



MillPlus IT

**NC Software
V5.20**

Programming- Manual

**V1.0
10/2003**

© HEIDENHAIN NUMERIC B.V. EINDHOVEN, THE NETHERLANDS 2003

The publisher accepts no liability in respect of specifications on the basis of the information contained in these instructions.

For the specifications of the numerical controller, please refer to the order data and corresponding specification description only.

All rights reserved. Copying of this manual or parts thereof only permitted with the written consent of the copyright holder.

Changes to and errors in this publication cannot be excluded. Therefore no claims can be laid to the information, figures and descriptions contained in this publication.

511 387-20

PS2773

TABLE OF CONTENTS

TABLE OF CONTENTS	I
1. INTRODUCTION	1
1.1 Foreword.....	1
1.2 Companion-manuals.....	1
2. IMPROVEMENTS IN V520 OVER ITS PREDECESSORS	3
2.1 Improvements implemented also in V510.	3
2.2 Improvements in V520.....	3
3. GENERAL PROGRAMMING INFORMATION	5
3.1 About partprograms	5
3.1.1 Program words	5
3.1.2 Program blocks.....	6
3.1.3 Writing a partprogram	7
3.1.4 Zero points	8
3.2 Axes configurations on machine tools	10
3.2.1 Defining coordinates	11
3.2.2 Cartesian-coordinates.....	11
3.2.3 Polar coordinates.....	11
3.2.4 Combining a linear coordinate and angle	12
4. ABOUT THIS MANUAL	13
4.1 Philosophy and purpose of the manual	13
4.2 Contents of each section	13
4.2.1 G-functions	13
4.2.2 F-functions	13
4.2.3 H-functions.....	13
4.2.4 M-functions	14
4.2.5 S-function.....	14
4.2.6 T-function.....	14
4.2.7 E-parameter	14
4.2.8 Geometric calculations with continuous movements.....	14
4.2.9 Graphical support	14
4.2.10 Machine constants.....	14
4.3 Programming functions.....	14
5. G-FUNCTIONS.....	17
5.1 G0 Rapid traverse.....	17
5.2 G1 Linear interpolation	20
5.3 G2/G3 Circular interpolation (CW/CCW)	27
5.4 G4 Dwell time	40
5.5 G6 Spline-interpolation	41

TABLE OF CONTENTS

5.6	G7 Tilting working plane	47
5.7	G8 Tilting tool orientation	54
5.8	G9 Define pole position (size reference point)	58
5.9	G11 Linear chamfer or rounding cycle	63
5.10	G14 Repeat function	70
5.11	G17 Mainplane XY, tool Z	72
5.12	G18 Mainplane XZ, tool Y	74
5.13	G19 Mainplane YZ, tool X	76
5.14	G22 Macro call	78
5.15	G23 Main program call	81
5.16	G25/G26 Enable/Disable feed- and/or speed-override	83
5.17	G27/G28 Positioning functions	84
5.18	G29 Jump function	87
5.19	G33 Basic Threadcutting movement	89
5.20	G36/G37 Activate/ Deactivate turning mode	89
5.21	G39 Activate/Deactivate tool offset	90
5.22	G40 Cancel tool radius compensation	92
5.23	G41/G42 Tool radius compensation (left/right)	95
5.24	G43/G44 Tool radius compensation to/past endpoint	103
5.25	G45 Axis parallel measuring movement and measuring tool dimensions	106
5.25.1	G45 Measuring a point	106
5.25.2	G45 + M25 Measure tool dimensions	110
5.26	G46 Measuring a full circle or probe calibration	112
5.26.1	G46 Measuring a full circle	112
5.26.2	G46 + M26 Probe calibration	115
5.27	G49 Checking on tolerances	117
5.28	G50 Processing measuring results	121
5.29	G51/G52 Cancel/activate pallet zero point shift	129
5.30	G53/G54—G59 Cancel/activate zero point shift	130
5.30.1	G53/G54—G59 Cancel/activate zero point shift (MC84=0)	130
5.30.2	G54 Extended zero point shift (MC84>0)	133
5.31	G61 Tangential approach	136
5.32	G62 Tangential exit	140
5.33	G63/G64 Cancel/Activate geometric calculations	144
5.33.1	Intersection point between two straight lines	147
5.33.2	Intersection point indicator	149
5.33.2.1	Intersection point between line and circle or circle and line	149
5.33.2.2	Intersection point between two circles	149
5.33.3	Programming a rounding	150
5.33.3.1	A rounding between intersecting line - circle or circle - line	150
5.33.3.2	A rounding between two intersecting circles	150
5.33.4	Two tangent geometry elements	151
5.33.4.1	Tangency indicator	151
5.33.5	Connecting circles	152
5.33.5.1	A connecting circle between line and circle or circle and line	152
5.33.5.2	A connecting circle between two circles outside each other	154
5.33.5.3	A connecting circle between two circles of which one circle inside the other one	154
5.33.5.4	A connecting circle between two concentric circles	155
5.33.6	Line definitions	156
5.33.6.1	Parallel line	158
5.33.6.2	Intersection point indicator	159
5.33.7	Continuous and non-continuous movement	161
5.34	G66/G67 Select negative/positive tool direction	162
5.35	G70/G71 Inch/Metric programming	164
5.36	G72/G73 Cancel/Activate scaling or mirror imaging	166
5.37	G74 Absolute position	171
5.38	G77 Bolt hole cycle	174
5.39	G78 Point definition	178
5.40	G79 Activate cycle	180
5.41	G81 Drilling cycle	183

5.42	G83 Deep hole drilling cycle	185
5.43	G84 Tapping cycle	188
5.44	G85 Reaming cycle	191
5.45	G86 Boring cycle	193
5.46	G87 Rectangular pocket milling	195
5.47	G88 Groove milling cycle	199
5.48	G89 Circular pocket milling cycle	203
5.49	G90/G91 Absolute/incremental programming	206
5.49.1	G90/G91 Absolute/incremental programming	206
5.49.2	Wordwise absolute and incremental programming	208
5.50	G92/G93 Incremental/Absolute zero point shift	209
5.51	G94/G95 Select feedrate unit	215
5.52	G96/G97 Constant cutting speed	217
5.53	G98 Graphic window definition	218
5.54	G99 Definition of workpiece blank as a box	220
5.55	G106 Kinematic Calculation: OFF	221
5.56	G108 Kinematic calculation: ON	222
5.57	G125 Lifting tool on intervention: OFF	224
5.58	G126 Lifting tool on intervention: ON	225
5.59	G136 Second axes configuration for fork head: ON	227
5.60	G137 Second axes configuration for fork head: OFF	230
5.61	G141 3D-Tool correction with dynamic TCMP	231
5.62	G145 Linear measuring movement	241
5.63	G148 Reading measure probe status	250
5.64	G149 Reading tool data or zero offset values	251
5.65	G150 Change tool data or zero offset values	254
5.66	G153 Correct workpiece zero point: OFF	256
5.67	G154 Correct workpiece zero point: ON	257
5.68	G174 Tool withdrawal movement	259
5.69	G180 Basic coordinate system	261
5.70	G182 Cylindrical coordinate system	263
5.71	G195 Graphic window definition	270
5.72	G196 End contour description	272
5.73	G197/G198 Begin inner/outer contour description	273
5.74	G199 Begin contour description	279
5.75	G200---G208 Pocket Cycle	285
5.75.1	G200 Begin pocket cycle	285
5.75.1.1	Introduction universal pocket cycle	285
5.75.1.2	Part program structure	287
5.75.1.3	Translation, rotation and mirror image of a pocket	288
5.75.1.4	Same pocket in another program	289
5.75.1.5	Operating section	290
5.75.1.6	Error messages	292
5.76	G201 Start contour pocket cycle	296
5.76.1	Usage of the generated macros	297
5.76.2	Macro for finishing a pocket contour	299
5.76.3	Sequence of the macros on the machine	301
5.77	G202 End contour pocket cycle	305
5.78	G203 Start pocket contour description	307
5.79	G204 End pocket contour description	309
5.80	G205 Start island contour description	310
5.81	G206 End pocket contour description	313
5.82	G207 Call island contour macro	315
5.83	G208 Quadrangle contour description	317
5.84	G217/G218 Deactivate/Activate angular head	321
5.85	G227/G228 Unbalance Monitor: ON/OFF	324
5.86	G240/G241 Contour check: OFF/ON	325
6.	SPECIFIC G-FUNCTIONS FOR MACROS	329

TABLE OF CONTENTS

6.1	Overview G-Functions for macros:	329
6.2	Error message functions	330
6.2.1	G300 Programming error messages	330
6.2.2	G301 Error in program or macro that just has been read in.	331
6.3	Executable functions	332
6.3.1	G302 Overwriting radius compensation parameters	332
6.3.2	G303 M19 with programmable direction	332
6.3.3	G310 Store table on disk	333
6.3.4	G311 Load table from disk	335
6.4	Query functions	336
6.4.1	G318 Read pallet or job table data	336
6.4.2	G319 Query actual technology data	336
6.4.3	G320 Query current G data	337
6.4.4	G321 Query tool data	342
6.4.5	G322 Query machine constant memory	343
6.4.6	G324 Query G-group	344
6.4.7	G325 Query M group	345
6.4.8	G326 Query actual position	346
6.4.9	G327 Query operation mode	347
6.5	Write functions	348
6.5.1	G331 Write tool data	348
6.6	Calculation functions	350
6.6.1	G341 Calculation of G7-plane angles	350
6.7	Formatted write functions	352
6.7.1	Introduction formatted write functions:	352
6.7.2	G350 Writing to a window	354
6.7.2.1	Writing to a window	354
6.7.2.2	Writing to a window and asking for information	355
6.7.3	G351 Writing to a file	356
6.8	Array functions	359
6.8.1	Introduction to array functions:	359
6.8.2	Overview array functions:	359
6.8.2.1	arrayNew (format)	360
6.8.2.2	arraySave (filename, internal array identification number)	360
6.8.2.3	arrayOpen (filename)	361
6.8.2.4	arrayExist (name)	361
6.8.2.5	arraySize (internal array identification number, rowcol)	361
6.8.2.6	arrayFind (internal array identification number, column, value)	362
6.8.2.7	arrayWrite (internal array identification number, row, column, value)	362
6.8.2.8	arrayRead (internal array identification number, row, column)	363
6.8.2.9	arrayFilter (name, column, criteria)	363
6.8.2.10	arraySort (name, column, order)	364
6.8.2.11	arrayDelete (name)	364
6.8.3	Method with Configuration file (previous versions)	365
7.	TOOL MEASURING CYCLES FOR LASER MEASURING	367
7.1	General remarks for laser measuring	367
7.2	G600 Laser: Calibration	369
7.3	G601 Laser: Measure tool length	371
7.4	G602 Laser: Measure length and radius	372
7.5	G603 Laser: Check of individual edge	374
7.6	G604 Laser: Tool breakage control	375
8.	MEASURING SYSTEM "TABLE-PROBE" (TT)	377
8.1	General notes measuring system "Table-Probe" (TT)	377
8.2	G606 TT: Calibration	378

8.3	G607 TT: Measuring tool length	379
8.4	G608 TT: Measuring tool radius	381
8.5	G609 TT: Measuring length and radius	383
8.6	G610 TT: Tool breakage control.....	385
8.7	G611 TT: Measuring turning tools	387
8.8	G615 Laser: Measuring turning tools	388
9.	MEASURING CYCLES	389
9.1	Introduction to measuring cycles	389
9.2	Description of addresses	390
9.3	G620 Angle measurement.....	392
9.4	G621 Position measurement	394
9.5	G622 Corner outside measurement.....	395
9.6	G623 Corner inside measurement	397
9.7	G626 Datum outside rectangle	399
9.8	G627 Datum inside rectangle	401
9.9	G628 Circle measurement outside	403
9.10	G629 Circle measurement inside	405
9.11	G631 Measure position of inclined plane	407
9.12	G633 Angle measurement 2 holes	409
9.13	G634 Measurement center 4 holes	411
9.14	G640 Locate table rotation center.	413
9.15	G642 Laser: Temperature compensation.....	416
10.	SPECIFIC CYCLES.....	419
10.1	G691 Measure unbalance.	419
10.2	G692 Unbalance checking.	419
10.3	G699 ATC- Cycle (= Application Tuning Cycle)	420
11.	MACHINING AND POSITIONING CYCLES.....	421
11.1	Summary of machining and positioning cycles:.....	421
11.2	Introduction	422
11.3	Description of addresses	423
11.4	G700 Facing cycle	424
11.5	G730 Multipass milling	426
11.6	G771 Machining on a line	428
11.7	G772 Machining on a rectangle.....	429
11.8	G773 Machining on a grid	430
11.9	G777 Machining on a circle	431
11.10	G779 Machining at a position	433
11.11	G781 Drilling / centring	434
11.12	G782 Deep hole drilling	435
11.13	G783 Deep drilling (chip breaking)	438
11.14	G784 Tapping with compensating chuck.....	440
11.15	G785 Reaming	442
11.16	G786 Boring	443
11.17	G787 Pocket milling	445
11.18	G788 Key-way milling.....	447
11.19	G789 Circular pocket milling.....	449
11.20	G790 Back-boring	451
11.21	G794 Interpolated tapping	453
11.22	G797 Pocket finishing.....	455
11.23	G798 Key-way finishing.....	457

TABLE OF CONTENTS

11.24	G799 Circular pocket finishing	459
12.	CYCLES IN THE G800 SERIES (TURNING).....	461
12.1	General description.....	461
12.2	G822 Clearance axial.....	461
12.3	G823 Clearance radial.....	461
12.4	G826 Clearance axial finishing.....	461
12.5	G827 Clearance radial finishing.....	461
12.6	G832 Roughing axial.....	461
12.7	G833 Roughing radial.....	461
12.8	G836 Roughing axial finishing.....	461
12.9	G837 Roughing radial finishing.....	461
12.10	G842 Grooving axial.....	461
12.11	G843 Grooving radial.....	461
12.12	G844 Grooving axial universal.....	461
12.13	G845 Grooving radial universal.....	461
12.14	G846 Grooving axial finishing.....	461
12.15	G847 Grooving radial finishing.....	462
12.16	G848 Grooving axial universal finish.....	462
12.17	G849 Grooving radial universal finish.....	462
12.18	G850 Undercut (DIN 76).....	462
12.19	G851 Undercut (DIN 509 E).....	462
12.20	G852 Undercut (DIN 509 F).....	462
12.21	G861 Threadcutting axial.....	462
12.22	G862 Threadcutting taper.....	462
13.	CYCLES IN THE G900 SERIES.....	463
13.1	General description.....	463
13.2	G951 Calibration.....	463
13.3	G953 Measure tool length.....	463
13.4	G954 Measure length, radius.....	463
13.5	G955 Cutter control shank.....	463
13.6	G956 Tool breakage control.....	463
13.7	G957 Cutter control shape.....	463
13.8	G958 Tool setting length, radius, corner radius.....	463
14.	CYCLE DESIGN	465
14.1	Introduction Cycle Design	465
14.2	Description of G function and addresses (G5?? CFG)	467
14.2.1	Example- G5?? CFG file (definition G5?? CFG)	467
14.2.2	Example-G550.CFG file	470
14.2.3	Permitted addresses	471
14.3	Support graphics.....	471
14.3.1	Making graphics in *.BMP format	471
14.3.2	Making graphics in *.DXF and *.PIC format	472
14.4	Execution macro	473
14.4.1	Example of execution macro	473
14.4.2	Explanation.....	473
14.5	Reading cycle files into the CNC.....	474
15.	TECHNOLOGICAL INSTRUCTIONS.....	475

15.1	F, F3=, F4= Feed and direction of the movement:	475
15.2	F1= Constant cutting feed by radius compensation of circles	476
15.3	F2=, F3=, F4= Feed in cycles	477
15.4	F5= Feed unit for rotary axes	478
15.5	F6= Local feed	478
15.6	H Auxiliary function	479
15.7	S-function	480
16.	M FUNCTIONS	481
16.1	M0/M1 Program stop	481
16.2	M3/M4/M5 Spindle-rotating clockwise/counter clockwise or spindle stop	482
16.3	M6 Automatic tool change	483
16.4	M7/M8/M9/M13/M14 Switch on/off coolant supply nr 2 / nr. 1	485
16.5	M19 Oriented spindle stop	486
16.6	M25 Measuring tool sizes	487
16.7	M26 Calibration the measuring probe	488
16.8	M24/M27/M28 Switch on/off a measuring probe	489
16.9	M30 End of partprogram	490
16.10	M41/M42/M43/M44 Select spindle speed range	491
16.11	M66 Manuel tool change	492
16.12	M67 Change tool values	493
17.	T-FUNCTION TOOL NUMBER AND TOOL MEMORY	495
17.1	T-function for tool change	495
17.1.1	Tool life monitoring	497
17.1.2	Tool breakage monitoring	497
17.1.3	Cutting force monitoring (T1=)	498
17.2	Tool memory	499
18.	E-PARAMETERS AND ARITHMETIC FUNCTIONS	503
18.1	E-Parameter	503
18.2	Arithmetical functions	505
18.2.1	Arithmetical operations	506
18.2.2	Trigonometrically and inverse trigonometrically functions	509
18.2.3	Relational expressions	509
18.2.4	Parentheses	510
19.	TURNING	513
19.1	Introduction	513
19.2	Machine constants	514
19.3	G36/G37 Switching turning mode on and off	515
19.4	G17/G18: Machining planes for turning mode	516
19.5	G33 Thread cutting	517
19.6	G94/G95 Expanded choice of feed unit	519
19.7	G96/G97 Constant cutting speed	520
19.8	Turning tools in the tool table	521
19.9	G302 Overrule radius comp. parameters	525
19.10	G611 TT130: Measure turning tools	526
19.11	G615 laser system: L/R measurement of turning tools	528
19.12	Unbalance cycles	530
19.12.1	General information	530

TABLE OF CONTENTS

19.12.2	Description of unbalance.....	530
19.12.3	(G227/G228) Unbalance monitor	531
19.12.4	G691 Measure unbalance.....	532
19.12.5	G692 Unbalance checking	534
19.13	Turning cycles	535
19.13.1	G822 Clearance axial.....	536
19.13.2	G823 Clearance radial	537
19.13.3	G826 Clearance axial finishing	538
19.13.4	G827 Clearance radial finishing	539
19.13.5	G832 Roughing axial	540
19.13.6	G833 Roughing radial	541
19.13.7	G836 Roughing axial finishing	542
19.13.8	G837 Roughing radial finishing	543
19.13.9	G842 Grooving axial.....	544
19.13.10	G843 Grooving radial	545
19.13.11	G844 Grooving universal axial roughing.....	546
19.13.12	G845 Grooving universal radial roughing	547
19.13.13	G846 Grooving axial finishing	548
19.13.14	G847 Grooving radial finishing.....	549
19.13.15	G848 Grooving universal axial, finishing.....	550
19.13.16	G849 Grooving universal radial, finishing	551
19.13.17	G850 Undercut DIN76.....	552
19.13.18	G851 Undercut DIN 509 E	553
19.13.19	G852 Undercut DIN 509 F	554
19.13.20	G861 Threading axial.....	555
19.13.21	G862 Treading conical.....	556
19.14	Examples.....	557
19.15	Survey of permitted G-Functions in the turning mode.	559

20. G64 GEOMETRIC CALCULATIONS WITH CONTINUOUS MOVEMENTS561

20.1	Conventions with the formats.....	561
20.2	Intersection point	563
20.2.1	Intersection point of two straight lines.....	563
20.2.2	Intersection point programmed as end point.....	566
20.2.3	Chamfer between intersecting straight lines	568
20.2.4	Rounding between intersecting straight lines	569
20.2.5	Rounding between straight line and chamfer.....	570
20.2.6	Intersecting point between line circle	571
20.2.7	Intersecting point of line and circle programmed as end point.....	573
20.2.8	Rounding between intersecting line and circle.....	575
20.2.9	Intersecting point between circle and line	577
20.2.10	Intersecting point of circle and line programmed as end point	579
20.2.11	Rounding between intersecting circle and line.....	581
20.2.12	Intersecting point between two circles	583
20.2.13	Intersection point between two circles programmed as end point	584
20.2.14	Rounding between two intersecting circles	586
20.3	Point of tangency.....	587
20.3.1	Point of tangency indicator (R1=).....	587
20.3.2	Tangent line and circle	587
20.3.3	Continuous connecting circle between tangent line and circle	590
20.3.4	Tangent circle and line	591
20.3.5	Continuous connection circle between tangent circle and line	593
20.3.6	Tangent circle and line	594
20.3.7	Continuous connecting circle between two tangent circles.....	595
20.4	Continuous connecting circle between elements which do not meet	596
20.4.1	Line and circle	596
20.4.2	Circle and line.....	598
20.4.3	Two circles outside each other.....	599

20.4.4	One circle inside the other one	600
20.4.5	Concentric circles	601
20.5	G64 Geometric calculations with non-continuous movements.....	602
20.5.1	Rounding or connecting circle indicator (K1=).....	602
20.5.2	Rounding with intersection points.....	602
20.5.3	Rounding between intersecting straight lines.....	603
20.5.4	Rounding between intersecting line and circle	605
20.5.5	Rounding between intersecting circle and line	606
20.5.6	Rounding between two intersecting circles	607
20.5.7	Tangent lines (R1=)	608
20.5.8	Connecting circle between a line tangent to a circle or v.v.	609
20.5.9	Connecting circle between a line which does not meet a circle	610
20.5.10	Connecting circle between circles outside each other	612
20.5.11	Connecting circle between two circles one inside the other.....	614
20.5.12	Connecting circle with two concentric circles	615
20.6	Examples	616
21.	APPENDIX	627
21.1	Tilting of the operating plane	627
21.1.1	Introduction	627
21.1.2	Machine types.....	628
21.1.3	Kinematics model	629
21.1.4	Operations	630
21.1.4.1	Manual operations	630
21.1.4.2	Display	630
21.1.4.3	Axis selection/position axis.....	631
21.1.4.4	Reference point	631
21.1.4.5	Intervention	631
21.1.5	Error messages	631
21.1.6	Machine Constants.....	632
21.2	Look Ahead Feed (LAF) function.....	633
21.2.1	Introduction	633
21.2.2	Detailed specification.....	633
INDEX	635

1. Introduction

1.1 Foreword

This manual assists you in programming the controller.

The machine should not be operated, even for a short period, by anyone who has not received the necessary training either in the Company, at an Institute of Further Education or in one of the Training Centres.

Please follow this advice to ensure proper usage.

The controller and the machine are coordinated using machine constants. Some of these constants are accessible to the user. Caution!

A thorough understanding of the significance and functions of these constants is required if they are to be changed. If in doubt please contact our Customer Service Department.

Users should therefore always output their programs and specific data (e.g. technical data, machine constants etc.) on their PC or onto diskette. This prevents data from being lost irretrievably if the battery or back-up battery is defective.

We reserve the right to change the design, equipment and accessories in the interest of further development. No liability will be accepted for any errors in the data, illustrations or descriptions.

1.2 Companion-manuals

The information relating to the installation, interfacing, operation, and programming for the controller cannot be adequately described in a single manual. Therefore, several manuals have been designed to give the user information relating to a particular type of task. The set of manuals available for the controller is listed in this section.

- User Manual
- CDS Manual (CNC Data Station Manual)
- Technical Manual
 - Some specific G-functions are describes in the Technical Manual.
- Interfacing
 - MIPS (Machine Interface Programming System.)
 - Basic IPLC Program

2. Improvements in V520 over its predecessors

2.1 Improvements implemented also in V510.

Added functions:

G125 Lifting tool on intervention: OFF	from V510_00b
G126 Lifting tool on intervention: ON	from V510_00b
G642 Laser: Temperature compensation	from V511_00

Modified functions:

G7/G8 Address L2= added.
G108 Without IPLC- shifts
G640 small changes.
G145 Address I4= air supply added.
G241 I1= Reverse check changed.
G320 extended with I1=66 to 73
Formatted wite functions
Extended with dependency condition (IF)
G787/G789/G797/G799 R1=67% replaced by R1=80%

Text changes:

G28 Acceleration reduction is not I6=0, but I6=100%.
G329 and G339 describes in Technical Manual
G786 I1= Address description changed.
G797/G799 B3=, I3= Address description added
Cycle Design.. The compression is further explained.

2.2 Improvements in V520.

Added functions:

G136 Second axes configuration for fork head: ON.
G137 Second axes configuration for fork head: OFF.
G153 Correct workpiece zero point: OFF
G154 Correct workpiece zero point: ON
G217 Deactivate angular head
G218 Activate angular head
G310 Store table on disk
G311 Load table from disk
G318 Read pallet or job table data
Array functions

Measurement cycles in main planes

G633 Angle measurement 2 holes
G634 Measurement center 4 holes
G699 ATC-Zyklus (= Applikation Tuning Cycle)

Turning cycles

G844 Universal grooving axial roughing
G845 Universal Grooving radial roughing
G848 Universal Grooving axial roughing
G849 Universal Grooving radial roughing
G850 Undercut (DIN 76).

- G851** Undercut (DIN 509 E).
- G852** Undercut (DIN 509 F).
- G861** Treadcutting cylinder.
- G862** Treadcutting taper.

Laser measurement cycles:

- G951** Calibration.
- G953** Measure tool length.
- G954** Measure length, radius.
- G955** Cutter control shank
- G956** Tool breakage control.
- G957** Cutter control shape.
- G958** Tool setting length, radius, corner radius.

Modified functions:

G240/G241 Calculating in advance of the contour with radius compensation (maximum 400 blocks)

G320 extended with I1=74 to 88

G350 extended with I2

Programming accuracy: The number of digits behind the decimal point is depending of MC705. MC705 can be 3 (accuracy 1µm or 1mGrad) or 4 (accuracy 0.1µ or 0.1mGrad).

Cycle Design: Adapted to programming-accuracy. Parameter INCH removed, FORM extended and DIMENSION added.

3. General programming information

3.1 About partprograms

A partprogram is the complete set of data and instructions required for producing a particular workpiece on a numerically controlled machine tool.

The instructions may contain different operations, such as milling, drilling, tapping, etc. Each separate operation is a unit, which can be split up into smaller instructions. One program block specifies one complete operation. The words in a block define the smaller instructions.

The proper machining sequence, with all the separate instructions, must be stated in a partprogram. Examples of separate instructions are tool movements, machine tool functions and technological data.

A program cannot be executed until it has been properly stored in the CNC system memory. A partprogram can be created and stored into the CNC memory in different ways:

1. Use interactive contour programming (ICP) for complex contours.
2. Use interactive partprogramming (IPP) for programming without knowledge of DIN programming.
3. Enter the program manually via the control panel.
4. Create the program separate from the control, use data terminal equipment to produce a data carrier (such as a punched paper tape, a magnetic digital cassette or disk) and input the data into the CNC memory.
5. By using networks facilities (for example: Ethernet or external Personal Computers).

3.1.1 Program words

The CNC PILOT control system employs the standard WORD ADDRESS system in which a word has two parts:

1. The addresses, which can be a single address (one alpha character) or an indexed address. An indexed address has an alpha character followed by an index and the character =, e.g. E1=.
2. A multi-digit number.

Words do not need leading zeros. However, if the value of a word is zero, then at least one zero must be written.

Format for words stating dimensional information, for example B, X, Y, Z, A, B, C and so on.

The words stating dimensional information can have a plus or a minus sign. If no sign is programmed, a positive value is assumed. A negative value must have a minus sign.

Dimension words can be written with a decimal point; therefore trailing zeros need not be stated. The control system assumes that the decimal point is behind the last digit of the number if the decimal point is not stated.

The number of digits behind the decimal point is depending of MC705. MC705 can be 3 (accuracy 1µm or 1mGrad) or 4 (accuracy 0.1µ or 0.1mGrad).

9 digits is always the total length. The programming is then 123456.789 or 12345.6789

Mm or Inch.

When G70 is programmed in the front of a program, the dimension is changed over to Inch. The programming of dimension words is then 12345.6789 or 1234.56789 (accuracy 0.0001 or 0.00001Inch)

Modal words

A modal word stays active after the execution, until the word is used again or reset.

Non-modal words

A non-modal word is only active in the block. When necessary it must program again.

Example of a single and an indexed word

Single word: X-21.43 'X' is the address, '-' is the sign and '21.43' the decimal number.

Indexed word: X1=-21.43 'X1=' is the address, '-' is the sign and '21.43' the decimal number.

3.1.2 Program blocks

A block can include several words considered as a unit, which contains all the information needed for one complete operation or function. This operation can be a tool movement or a machine tool function, or a combination of both.

The CNC PILOT control system employs a VARIABLE BLOCK FORMAT. The block lengths can be different because of changes in the number or length of the words. A block can contain up to 255 characters.

The N-word must always be the first word in a block. The other words can be written in any order. The example gives the preferred sequence for the frequently used words.

Each word can occur only once in a block. Words such as E1= and E2= have different addresses and therefore can be both present in the same block.

On a data carrier the character line feed [LF] separates the blocks.

Example of a program block

N20 G1 X14 Z62.5 F300 S200 T12 M3	N20: Block number
	G1: Preparatory function
	X14 and Z62.5: Dimensional information

Technological and machine data such as spindle speed (S), feedrate (F), tool selection (T) and E.g. a direction of spindle rotation (M3) may be included as well.

The block number N

The first word in a block is the block number, which identifies that block. Each block must have a separate number. The block numbers range from N0 to N9999999.

A general rule is that a block number cannot be in the same program more than once. However, the check on the block numbers is inactive if a machine constant is set or the BTR possibility is used. The machine constant setting is useful when large programs should be executed and the BTR possibility is not used.

Block numbers can be in any sequence. The execution will be in the programmed sequence e.g.

Programmed sequence: N10, N50, N30
 Executed sequence: N10, N50, N30

The re-number function of the control allows the block numbers to be automatically renumbered in increasing order, starting from N1.

The CNC system automatically generates block numbers when the programmer uses the control panel to input programs.

3.1.3 Writing a partprogram

Program identification

Each partprogram or subprogram has to start with an identification number, which ranges from 1 to 9999999, dependent of machine constant MC773.

So numbers as 1, 125, 9001, 12345, 876543, 3451592 are valid identification numbers.

The rename function of the control is available for changing the identification number.

A partprogram name can be written between the characters CONTROL OUT '(' and CONTROL IN ')' and immediately after the identification number. These names are listed if the file directory is displayed on the control.

Example of partprogram identification with name
9001 (PLATE NR. A334)

The following partprogram identification is possible for earlier CNC systems (compatibility) (for programs %PM... and macros %MM...)

%PM9001
N9001 (PLATE NR. A334)

These programs are automatically identified and stored correctly by the CNC system, dependent of syntax check (MC772). Data transmission from the CNC to the outside is controlled by machine constant (MC799)

Partprogram setup

To write a partprogram the programmer must do the following:

1. Determine the mounting of the workpiece and the position of the clamps
2. Determine the machining operation sequence
3. Determine the tools required for the operations
4. Determine for each tool the appropriate technological data
5. Determine the workpiece dimensions and the necessary movements.

The points 1 to 4 are outside the scope of this manual.

The movements on the machine are a combination of tool and workpiece movements. To simplify the programming the programmer should assume that all movements are tool movements. The configuration of the machine tool and CNC system determines how the movements are actually performed.

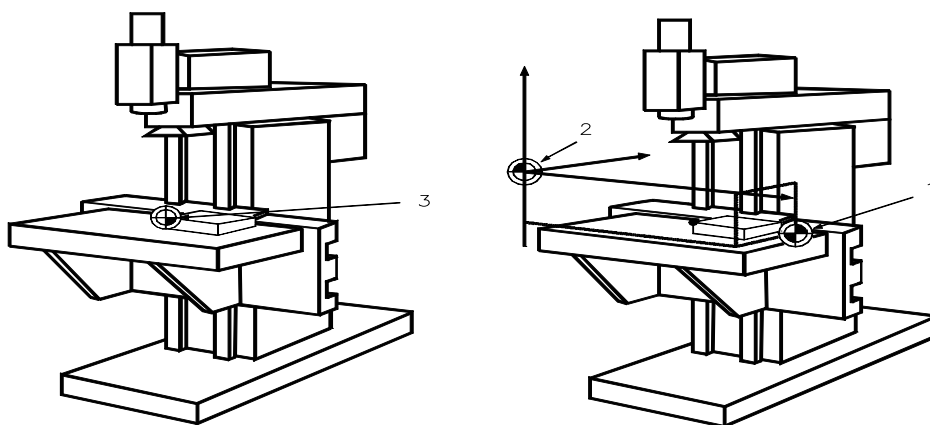
An imaginary coordinate system is positioned on the workpiece so that programmed movements refer to a zero. The programmer determines the position of this point in such a manner that the easiest programming calculations are produced. Refer to Axes configurations on machine tools for the directions of the coordinate axes.

Program storage

The user memory can store two partprogram (execution and editing) and several subprogram (macro). All other programs and macros will be stored on the disc. A machine constant (MC85) sets the maximum amount of programs to be between 50 and 1000.

With the lock function it is possible to protect partprograms and macros against unauthorised editing on the control.

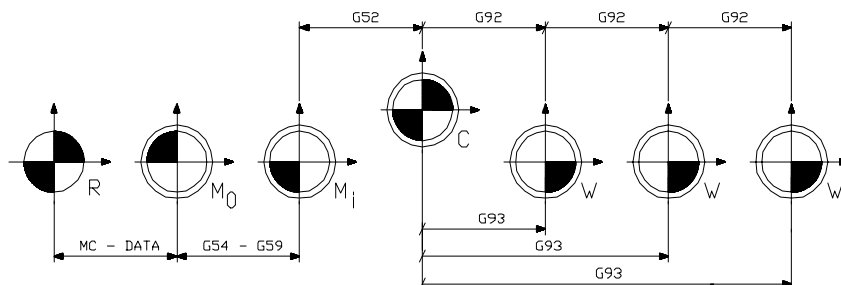
3.1.4 Zero points



- 1 = Machine reference point
- 2 = Machine zero point
- 3 = Program zero point

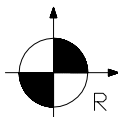
After power-on, REFERENCE POINT SEARCH must be carried out first. As a result, the machine zero point is known, since the zero offsets from the machine zero point (M0) to the machine reference point (R) are stored as machine constants.

The part programmer establishes a program zero point (W), which is related to the part and from which the part dimensions are measured. This program zero point must also be related to the machine zero point, which can be established with the functions G52 and/or G54-G59.



1. Machine reference point (R)

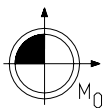
Each axis of a machine tool has a fixed point called the reference point of the axis. The reference points of all axes form the machine reference point (R).



During "reference point search" (refer to the Operating Manual) the tool moves to the reference point of the selected axis (or axes). When the reference point is reached, the axis is automatically zeroed by the control and the positions of the software limit switches are set.

2. Machine zero point (M0)

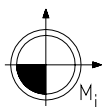
The machine zero point is also a fixed point on the machine.



When the CNC control system is commissioned, the distances from the machine reference point (R) to the machine zero point (M0) are measured along the axes and stored in the machine constant memory. Each axis has its own machine constant for this purpose.

After the machine reference point is established by REFERENCE POINT SEARCH, the control system reads the associated dimensions from the machine constant memory. The machine zero point (M0) is set as the origin of the coordinate system and the displayed positions are related to this zero point.

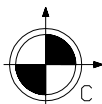
3. Secondary machine zero point (Mi)



When the machine tool has several clamping stations (e.g. pallet stations) each clamping station must have its own fixed zero point. These fixed zero points are called secondary machine zero points (Mi)

The zero offset memory contains the axis distances between the machine zero point (M0) and the secondary machine zero points (Mi). 6 or maximum 99 secondary points can be stored by using the G54 to G59, G54I[0..99](from V320) functions.

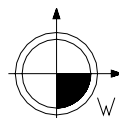
4. Mounting zero point (C)



When a secondary machine zero point (Mi) is established, the zero point of the mounting device must be determined. This zero point may coincide with the active Mi or can be set by the G52 PRESET AXIS function.

Zero point C is automatically set by the control, when an external program call with offset values is made

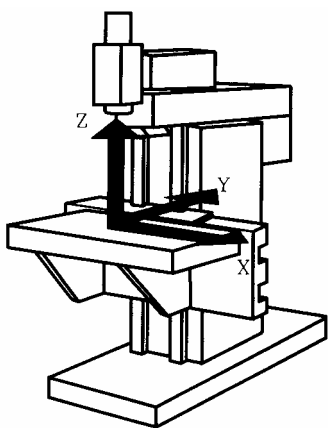
5. Program zero point (W)



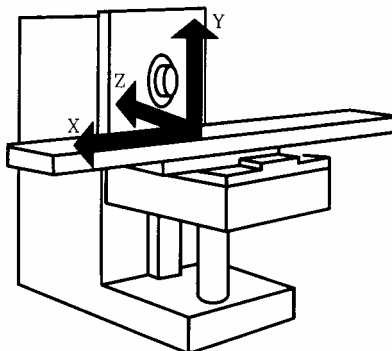
The program zero point W is the zero point from which the axis coordinates in the partprogram are measured. The programmer can set the position of point W arbitrarily.

The functions G52, G54-G59, G54I[0..99] and G92/G93 establish the relation between the program zero points and the machine zero point.

3.2 Axes configurations on machine tools

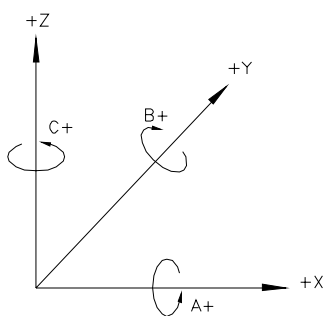


Vertical knee milling machine

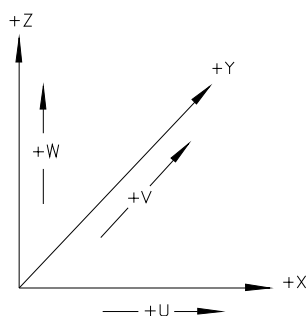


Horizontal knee milling machine

A milling machine has three main linear axes (X, Y, Z), which are at 90° to each other. The orientation of these axes is established by the Z-axis, which is always parallel to the main spindle of the machine tool. The X-axis is horizontal and parallel to the work holding surface. Each main axis can have a rotary axis and a linear axis, parallel to a main axis. These are shown in the illustration below.



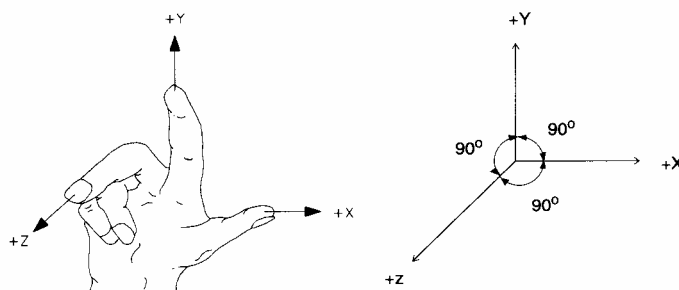
Rotary axes



Linear axes parallel to main axes

Orientation of main axes, rotary axes and linear parallel axes.

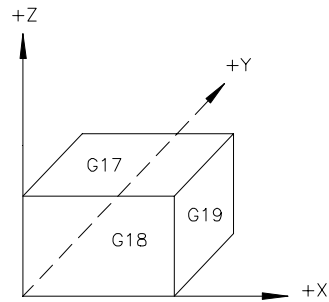
The standards ISO 841, DIN 66217 and EIA RS-267-A, all define the positions of axes on a numerically controlled machine. The right-hand rule is used for stating the orientation for all CNC machines axes.



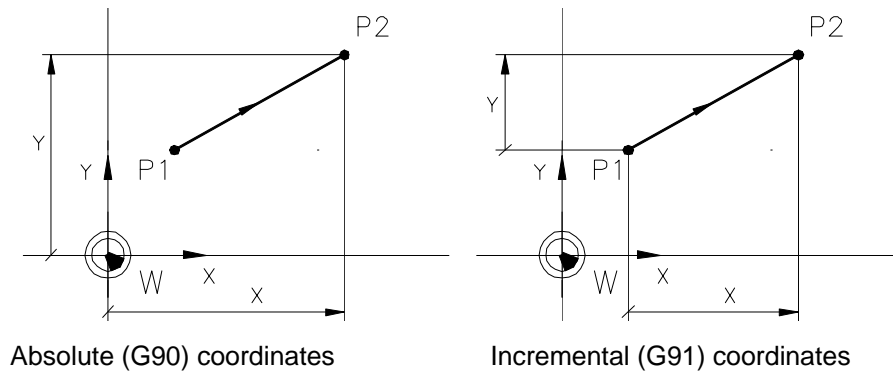
The thumb represents the X-axis, the forefinger the Y-axis and the middle finger the Z-axis. The directions in which the fingers are pointing represent the positive directions along the axes.

3.2.1 Defining coordinates

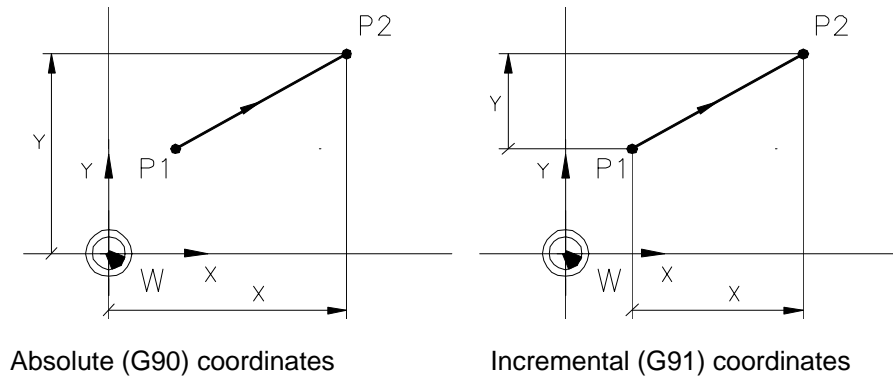
Axes distances define the coordinates of points in three-dimensional (3-D) space. Axis coordinates will be in one of three planes (XY-plane, XZ-plane, YZ-plane).



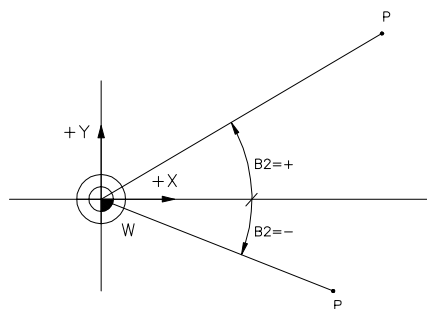
3.2.2 Cartesian-coordinates



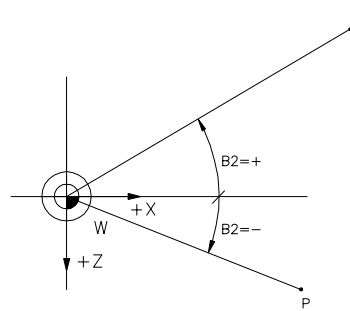
3.2.3 Polar coordinates



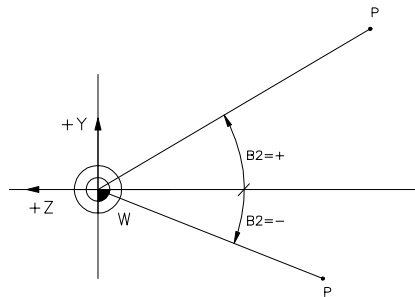
XY-plane (G17)



XZ-plane (G18)

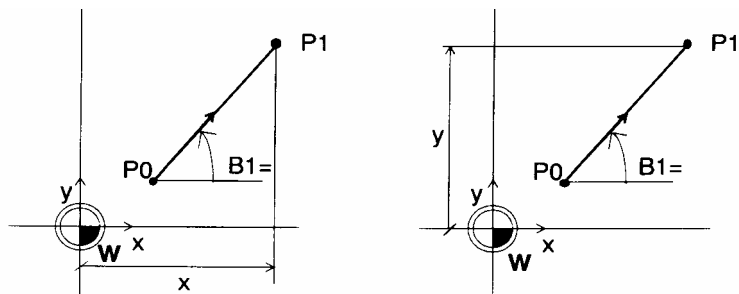


YZ-plane (G19)

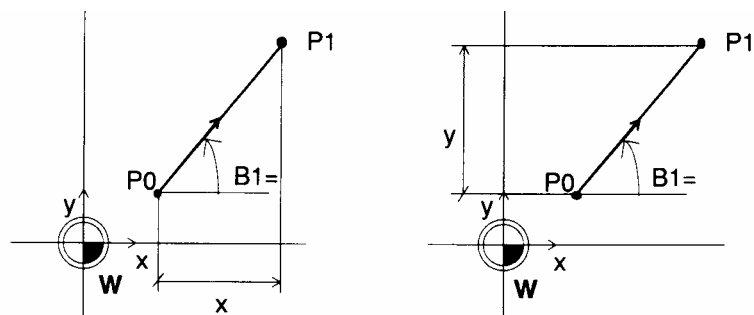


3.2.4 Combining a linear coordinate and angle

One linear axis coordinate and an angle can, in combination, define a point's position.



Absolute (G90) coordinates



Incremental (G91) coordinates

4. About this manual

4.1 Philosophy and purpose of the manual

This manual has been arranged to allow access to comprehensive information relating to programs for the CNC system.

The core of the reference data in this manual is contained within the sections, which describe the F, G, H, M, S and T- functions. Additional information, such as mathematical operations and formula, is contained in appendices.

4.2 Contents of each section

4.2.1 G-functions

G-functions are functions preparing the CNC controlled machine to programming instructions and are therefore named Preparatory functions. The contents of each section describing a G- function is placed under standard headings, which are:

Purpose

The reason(s) for using the function

Format

The format(s) of a program block containing the function. The conventions used under this heading are given in the Introduction section of this manual.

Parameters

Program words defining the extent of the function's influence, or words, which can be programmed, when the function is active.

Associated functions

Functions, which are in the same functional group, they can cancel the function.

Type of functions

Explains if the function is modal or not.

Notes and usage

Explanation(s) of how the function is used and under what circumstances.

Examples

Examples showing practical applications of the function.

4.2.2 F-functions

F-functions are functions, which establish the feedrate (mm/min or inch/min). The feedrate is dependent of the situation. See chapter: Technological instructions.

4.2.3 H-functions

H-functions are assigned tasks by the machine tool builder. The programmer must therefore read the machine tool builder's documentation for description(s) of how this auxiliary function is used. See chapter: Technological instructions, paragraph: H Auxiliary functions.

4.2.4 M-functions

M-functions are functions, which directly affect CNC machine operations, e.g. switching the coolant supply on and off. The programmer must therefore read the machine tool builder's documentation for description(s) of how this auxiliary function is used. See chapter: M functions.

4.2.5 S-function

The S-function specifies the spindle speed in revolutions per minute (RPM).
See chapter: Technological instructions, paragraph: S function.

4.2.6 T-function

The T-function specifies the number, which is used to select a tool and also to store its dimensions in the CNC's Tool Memory. See chapter: T function tool number and tool memory.

4.2.7 E-parameter

E-parameter is useful in making a program more flexible. One program can be used for different products. See chapter: E-parameters and arithmetic functions.

4.2.8 Geometric calculations with continuous movements

Geometric calculations with continuous movements is a function which able the operator to make a program without knowing the exact coordinates of a certain point. See: G64.

4.2.9 Graphical support

Graphical support visualizes the movement of the tool in different ways on the display.

4.2.10 Machine constants

With machine constants the machine manufacturer can customize the controller for his machines. The operator can change some specific machine constants (Operator MC). For description machine constants refer to the technical manual.

4.3 Programming functions

Fundamentals of CNC Programming

Coordinate Measurement Modes

- | | |
|---------|-------------------------------------|
| G90/G91 | - Absolute/incremental programming. |
| G70/G71 | - Inch/metric programming. |

Basic Tool Movements

- | | |
|-------|---|
| G0 | - Rapid traverse. |
| G1 | - Linear interpolation. |
| | - Tool movement with a feedrate (in a linear and a rotary movement) |
| | - Tool movement with a feedrate (3D-interpolation) |
| G2/G3 | - Circular interpolation (CW/CCW). |
| G78 | - Point definition. |

Radius Compensation

- | | |
|---------|--|
| G41/G42 | - Tool Radius Compensation (Left/Right). |
| G43/G44 | - Tool Radius Comp. TO/PAST End Point. |
| G40 | - Cancel Tool Radius Compensation. |

Main Planes

G17	- Main plane XY, tool Z.
G18	- Main plane XZ, tool Y
G19	- Main plane YZ, tool X
G7	- Tilt operating planes

Positioning & Feedrate Functions

G27/G28	- Cancel/activate positioning function.
G25/G26	- Feed override active/inactive.
G94/G95	- Select feedrate unit.
G4	- Dwell time.

Tool & Spindle Speed Functions

S	- Spindle speed.
T	- Tool number.

Zero Datum Points

G51/G52	- Cancel/activate G52 zero point shift.
G53/G54-G59	- Cancel/activate zero point shift
G54 I[0..99]	- Activate zero point shift
G92/G93	- Incremental/absolute zero point shift.

Graphical Simulations

G98	- Graphic window definition.
G99	- Definition of workpiece blank as a box.
G195	- Graphic window definition.
G196	- End contour description.
G197	- Begin inner contour description.
G198	- Begin outer contour description
G199	- Begin contour description.

Machine Functions

M3/M4	- Spindle clockwise/counter-clockwise.
M5	- Spindle stop
M19	- Orientated spindle stop
M7	- Switch on number 2 coolant supply.
M8	- Switch on number 1 coolant supply.
M9	- Switch off coolant supply.
M13	- Switch on No. 1 coolant rotate spindle clockwise.
M14	- Switch on No. 1 coolant rotate spindle counter-clockwise.
M6	- Automatic tool change.
M66	- Manual tool change
M67	- Change tool compensation values
M0	- Program stop.
M1	- Optional program stop.
M30	- Partprogram end.

Geometric Functions

G11	- Polar coordinate, corner rounding, chamfer.
G63/G64	- Cancel/activate geometric calculations.
G72/G73	- Cancel/activate scaling or mirror imaging.
G9	- Polar point definition

Defined (Canned) Cycles

G79	- Activate cycle.
G77	- Bolthole circle.
G81	- Drilling cycle.
G83	- Deep hole drilling cycle.
G84	- Tapping cycle.
G85	- Reaming cycle.

- | | |
|-----|-------------------------------------|
| G86 | - Boring cycle. |
| G87 | - Rectangular pocket milling cycle. |
| G88 | - Slot milling cycle. |
| G89 | - Circular pocket milling cycle. |

Transfer of Program Control

- | | |
|---------|---------------------------------------|
| G14 | - Repeat function. |
| G29 | - Conditional jump. |
| G22 | - Macro call. |
| G23 | - Program call. |
| G36/G37 | - Activate / deactivate turning mode. |

E-parameters and arithmetical operations

Special Functions

- | | |
|-----------|---|
| G141 | - 3D tool correction. |
| G180/G182 | - Cancel/activate cylinder interpolation. |
| G6 | - Spline interpolation. |
| G8 | - Swivel tool |
| G39 | - Activate/deactivate compensation |
| G61/G62 | - Tangential approach and exit |
| G66/G67 | - Select negative / positive tool direction |
| G74 | - Absolute position |
| G174 | - Tool withdrawal movement |

Measuring Cycles

- | | |
|---------|-------------------------------------|
| G45 | - Axis parallel measuring movement. |
| G45 M25 | - Measure tool dimensions. |
| G46 | - Measuring a full circle. |
| G46 M26 | - Probe calibration. |
| G49 | - Checking on tolerances. |
| G50 | - Processing measuring results. |
| G145 | - Linear measuring movement. |
| G148 | - Read probe status. |
| G149 | - Read tool data and offsets. |
| G150 | - Write tool data and offsets. |

Pocket cycle

- | | |
|------|------------------------------------|
| G200 | - Begin pocket cycle |
| G201 | - Start contour pocket cycle |
| G202 | - End contour pocket cycle |
| G203 | - Start pocket contour description |
| G204 | - End pocket contour description |
| G205 | - Start island contour description |
| G206 | - End island contour description |
| G207 | - Call island contour macro |
| G208 | - Quadrangle contour description |

Auxiliary Function

H

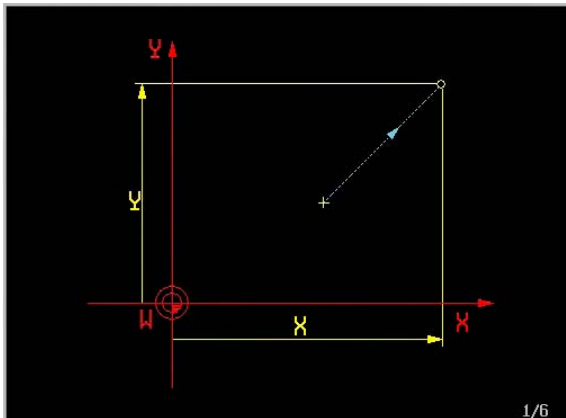
5. G-Functions

5.1 G0 Rapid traverse

To use the rapid traverse rate for axis movements. This traverse rate is set by Machine Constants (per axis). The function G0 is used mainly for positioning a tool before and after cutting passes.

Format

G0 [axis coordinates]



G Rapid traverse
 X Endpoint coordinate
 Y Endpoint coordinate
 Z Endpoint coordinate
 B Endpoint angle
 C Endpoint angle
 B1= Angle
 B2= Polar angle
 ?90= Endpoint abs. (X,Y,Z..)
 ?91= Endpoint incr. (X,Y,Z..)
 L1= Path length
 L2= Polar length
 P1= Point definition number

Notes and usage

Modality

This function is modal with G1, G2, G3, G6 und G9.

Modal words

F, F1=, F3=, F4=, S, S1=, T, T1=, T2=, T3=, M, M1=, H, Ennn, U, V, W
 D (Angle oriented spindle stop, only in combination with M19),
 I, J, K, R inside G182
 I, I1=, J1=, K1=, R, R1=, X1=, Y1=, Z1= inside ICP
 A40=, B40=, C40=, R for feed calculations axes.

Non modal words

F6=

Default mode

The G0 function is automatically set at the start of a program, after softkey <Clear control>, after softkey <Cancel program>, and after executing G77 or G79

Cancellation

The function G0 is cancelled by a G1, G2, G3, and G6.

Stop after a rapid movement

The programmed position is reached before the next movement starts. So a stop occurs after a rapid movement.

No stop after a rapid movement

If required, rapid movements can be executed without a stop. The function G28 and parameter I4= are used to state, that rapid movements are executed with (default) or without a stop. Refer to the G28 function "rapid traverse movements" for additional Information about G28 and I4=.

G0 RAPID TRAVERSE

Movements in the main plane

Rapid movements in the main plane, thus the plane defined with G17, G18 or G19, are executed under full control of the linear interpolator. So a straight line is made.

Polar coordinates or one coordinate and angle

Positions in the main plane can also be programmed with polar coordinates or one coordinate and angle.

Positioning logic

The positioning logic is a fixed sequence of axis movements depending on the active main plane and the movements along the tool axis.

When the tool is moving towards the workpiece:

	G17 XY-PLANE	G18 XZ-PLANE	G19 YZ-PLANE
1st movement	Rotary axes	Rotary axes	Rotary axes
2nd movement	X and Y	X and Z	Y and Z
3rd movement	Z-axis	Y-axis	X-axis

When the tool is moving away from the workpiece:

	G17 XY-PLANE	G18 XZ-PLANE	G19 YZ-PLANE
1st movement	Z-axis	Y-axis	X-axis
2nd movement	X and Y	X and Z	Y and Z
3rd movement	Rotary axes	Rotary axes	Rotary axes

Tool in positive direction of tool axis (G66/G67)

With G67 the tool is pointing in the positive direction of the tool axis, which means that a movement towards the work piece is in the positive direction. The positioning logic is changed accordingly.

Switching off positioning logic

Sometimes the positioning logic is not required, e.g. when moving the tool to a tool change position. If the positioning logic is switched off, all axes move simultaneously.

The positioning logic can be switched off with the G28 function and the word I5=1. Refer to G28 for details.

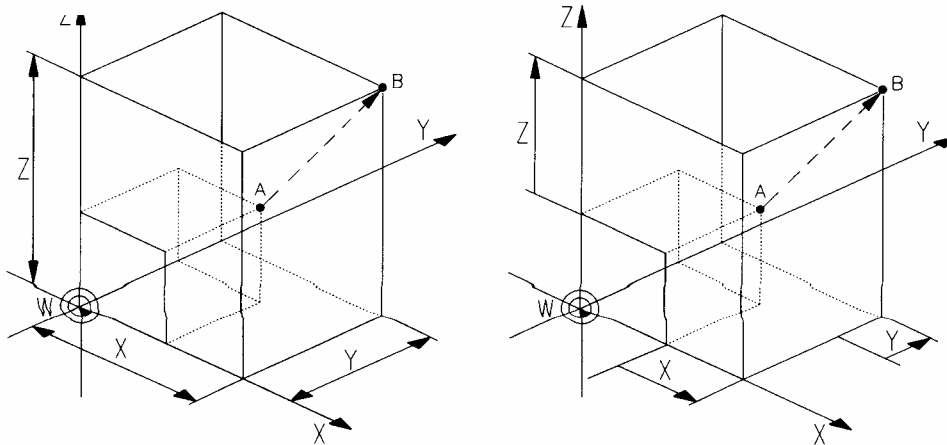
Oriented spindle stop (D.. M19)

The D-word for the offset angle with oriented spindle stop must be programmed together with the function M19. Refer to the function M19 for details.

See also G303 M19 D..

Starting position

At the beginning of a program, every active axis should be programmed in a program block for axis movements to ensure that each axis is in the starting position after a program start.

Example

The rapid movement from point A to point B can be programmed as:
N... G0 X25 Y15 Z30

The actual movements are (Bei G17/G66)

- a movement in the tool axis (Z).
- a simultaneous movement in the main plane (X and Y);

5.2 G1 Linear interpolation

Commands all linear movements to be at specified feedrates.

Format

Main plane movements

G1 {X..} {Y..} {Z..} {F..}

3D-interpolation

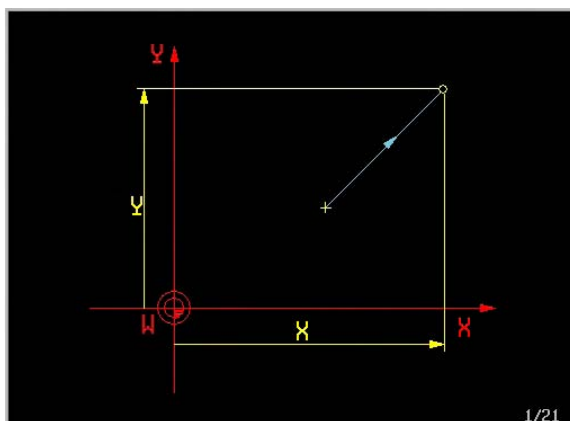
G1 X.. Y.. Z.. {F..}

One rotary axis

G1 {A..} {B..} {C..} {A40=..} {B40=..} {C40=..} {F...}

Multi axes movements

G1 {X..} {Y..} {Z..} {A..} {B..} {C..} {A40=..} {B40=..} {C40=..} {F..}



G Linear interpolation
 X Endpoint coordinate
 Y Endpoint coordinate
 Z Endpoint coordinate
 B Endpoint angle
 C Endpoint angle
 B1= Angle
 B2= Polar angle
 ?90= Endpoint abs. (X,Y,Z..)
 ?91= Endpoint incr. (X,Y,Z..)
 L1= Path length
 L2= Polar length
 P1= Point definition number
 P2= Point definition number
 P3= Point definition number

P4= Point definition number

Notes and usage

Modality

This function is modal with G0, G2, G3, G6 and G9.

Modal words

F, F1=, F3=, F4=, F6=, S, S1=, T, T1=, T2=, M, M1=, H, Ennn, U, V, W

D (Angle oriented spindle stop, only in combination with M19),

I, J, K, R inside G182

I, I1=, J1=, K1=, R, R1=, X1=, Y1=, Z1= inside ICP

A40=, B40=, C40=, R for feed calculations rotary axes.

Axis movements

Axis movements are always interpolated, so that they occur simultaneously along all the programmed axes.

Polar coordinates or one coordinate and angle

Positions in the main plane can also be programmed with polar coordinates or one coordinate and angle.

Defined points (G78)

A maximum of four previously defined points (P-words) can be stated in a G1 program block. The sequence in which the points are programmed also establishes the tool movement sequence e.g.:

G1 P5 P2 F100

Tool moves at 100 mm/min first to P5 and then to P2.

Feedrate in main plane

The programmed feedrate is the feed on the straight line.

Radius compensation in main plane (G40 - G44)

Radius compensation on contours defined with linear and circular movements is available. Refer to the functions G40, G41/G42 and G43/G44 for additional information.

3D-Interpolation

If the three axes X, Y and Z are programmed in one block, the CNC will control the axes movements so that a linear movement in space is made from the start point to the point defined by the three end point coordinates.

Positions in the main plane can be programmed normally.

For programming axes outside the main plane Cartesian coordinates must be used.

Feedrate with 3d-interpolation

The programmed feedrate is the feed on the straight line.

3D tool correction

A 3D tool correction with normalized vectors is available. Refer to the G141 section for additional information.

Programming rotary axes**Radius of rotary axis for feed calculations**

For feed calculations the radius of each rotary axis involved can be programmed with A40= (for the A-axis), B40= (for the B-axis) and C40= (for the C-axis).

Machines with kinematic model

In machines with the kinematical model, the radius of the axis of rotation between the centre point of the rotary axis and of the workpiece can be calculated. This means that A40=, B40= and C40= no longer need to be programmed. This is enabled via G94 F5=1.

Cancellation of the radius of the rotary axis

The programmed radius of the rotary axis is modal and, therefore, remains active until cancelled by:

- A40=0, B40=0 or C40=0.
- A different coordinate system being selected.
- M30, CLEAR CONTROL or CANCEL PROGRAM.

Programmed feedrate

When A40=, B40= or C40= is given a radius value, the surface feed is programmed in mm/min or inch/min.

No radius of rotary axis programmed

If no radii are programmed, the programmed feedrate is the feed of the path of the linear axes and is used by the CNC for calculating the feed for each rotary axis. This ensures that all axes cover the same part of their distances to go in the same time.

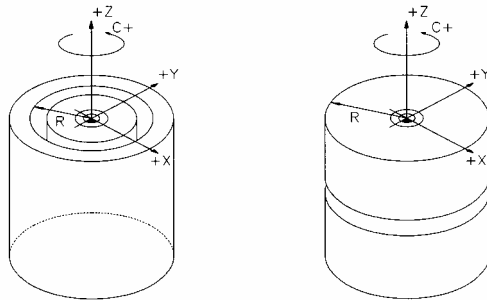
Maximum feed per axis

If the maximum feed of an axis (MC-setting) is exceeded, the actual feed is reduced, so that the movement is performed with the maximum feedrate.

One rotary axis

Note: In the following example, the Z- and C-axis have been used, however, the same principles apply to the Y-/B-axis and X-/A-axis combinations.

With just the rotary axis moving, two cutting actions are possible.



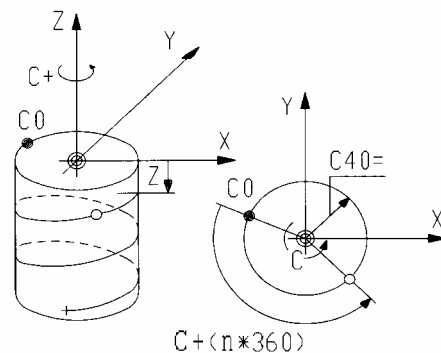
1. Groove cut in the facing plane

2. Groove cut in the cylinder's surface

Feedrate with rotary axis only

If the radius of the rotary axis is programmed (A40=..., B40=..., C40=...), the feedrate is the surface feed in mm/min or inch/min.

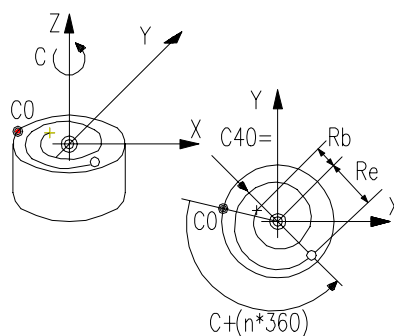
If the radius of the rotary axis is not programmed or A40=0, B40=0 and C40=0 is programmed, the feedrate is the feed in degrees/min.

One rotary axis and one linear axis

Helix on the curved surface of a cylinder

Cylinder interpolation

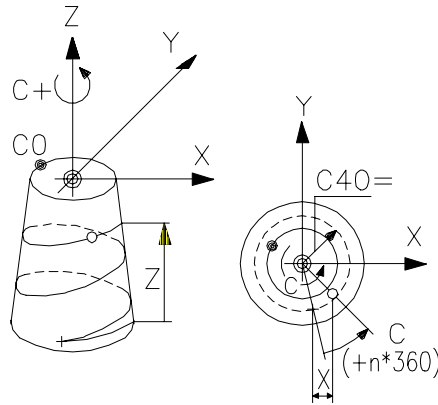
If contours or positions are to be programmed on the curved surface of a cylinder, the function G182 is available for an easy programming of these movements. Refer to the G180/G182 section for additional information.

Spiral in the facing plane

Average radius of the spiral

The radius value, which should be programmed for feed calculations, is the average radius of the path; this radius is used by the CNC to calculate the required feedrate to produce the spiral. The average radius (e.g. C40=) is calculated by using the formula:

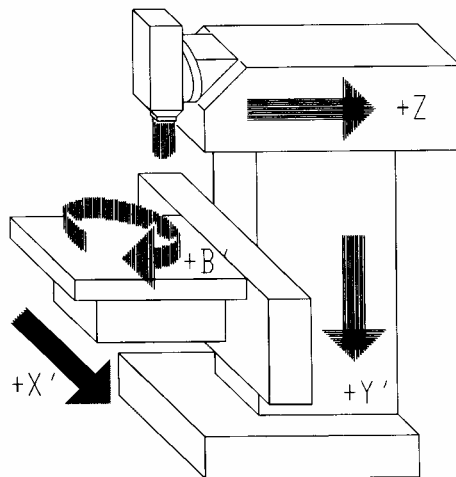
$$C40 = (\text{radius at start point (Rb)} + \text{radius at end point (Re)}) / 2$$

One rotary axis and two linear axes

For feed calculations the average radius (C40=) has to be used and is calculated as:

$$C40 = (\text{radius at start point} + \text{radius at end point}) / 2$$

The movement on the cone is programmed with all three axes (X, Z, and C) in one block.

Multi axes programming

Machine tool with rotary table and tilting head

In a G1-block any combination of the three linear axes X, Y, or Z and the three rotary axes A, B or C (if available) is allowed.

Positions in the main plane can be programmed normally.

For programming axes outside the main plane Cartesian coordinates must be used.

General remarks with G1**Parallel axes**

If available on the machine tool, the linear axes U, V and W, which are parallel to the main axes X, Y and Z can be used instead of X, Y and Z. Only Cartesian coordinates can be used with the U, V and W axes.

Cancellation

The function G1 is cancelled by functions G1, G2/G3, G6 or at end of program (M30), CLEAR CONTROL or softkey CANCEL PROGRAM.

Start next movement

In general feed movements are executed without a stop between the blocks. This results in rounded corners.

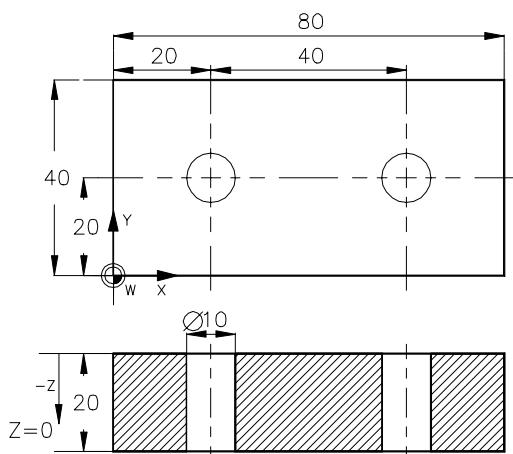
G28 and parameter I3= allows to program if the next movement starts after a full stop of the tool or without a stop between the movements.

Refer to the function G28 for details.

Oriented spindle stop (D..M19)

The D-word for the offset angle with oriented spindle stop must be programmed together with the function M19. Refer to the function M19 for details.

See also G303 M19 D..

Examples**example 1. Drilling**

N9001

N1 G17

N2 G54 T1 M6 S1000

N3 X20 Y20 Z1 F150 M3

N4 G1 Z-23.5

N5 G0 X60 Z1

N6 G1 Z-23.5

N7 G0 Z200 M30

Activate XY-plane (G17).

Activate zero shift (G54). Load tool T1 and its offsets. Spindle rotation at 1000 rev/min. Drill diameter 10 mm.

Move tool rapidly (G0) to programmed position. Set feedrate to 150 mm/min. Make spindle rotate clockwise (M3).

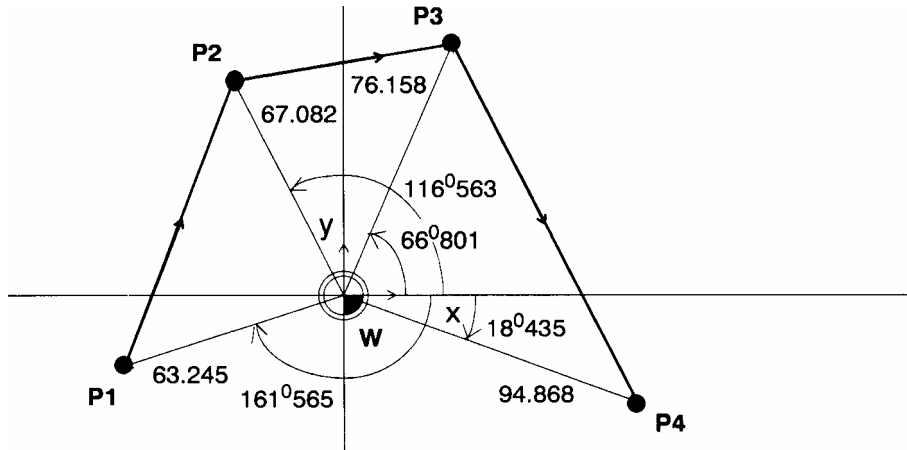
Feed tool to programmed depth.

Retract tool to Z1 and then move the tool rapidly to X60. The CNC's positioning logic ensures that the tool does not collide with the workpiece, because the tool is first moved along the Z-axis before moving along the X-axis.

Feed tool to programmed depth.

Retract tool to Z200 and end of program.

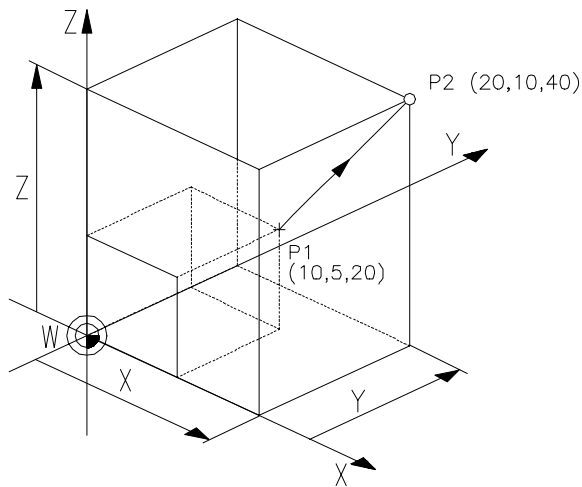
Example 2. Main plane movements



By using absolute polar coordinates the above movements can be programmed as follows:

N10 G0 B2=-161.565 L2=63.245 F100	(P1)	Move at rapid traverse rate (G0) to point P1.
N11 G1 B2=116.565 L2=67.082	(P2)	Move tool at feedrate (100 mm/min) to the points P2, P3 and P4.
N12 B2=66.801 L2=76.158	(P3)	
N13 B2=-18.435 L2=94.868	(P4)	

Example 3. 3D-interpolation

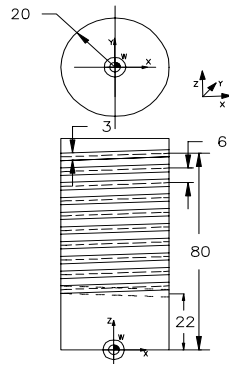


Tool moves from P1 (10,5,20) to P2 (20,10,40) at a feedrate of 100 mm/min

```
N14 G0 X10 Y5 Z20
N15 G1 X20 Y10 Z40 F100
```

In block N15 the three axes move simultaneously and reach their end positions at the same time.

Example 4. Spirale auf einer Zylinderfläche



Simultaneous movements of Z- and C-axes. The tool is in the Y-axis. The helix has 10 turns and a pitch of 6 mm.

N10 G18

Define the main plane

N11 T1 M6

Load tool T1 and its offsets (mill diameter 3 mm). Tool is in Y-axis (G18).

N12 G0 X0 Y22 Z80 C0 S3000 M3

Start spindle and move tool to start position.

N13 G1 Y18 F75

Feed tool 4 mm to position Y18

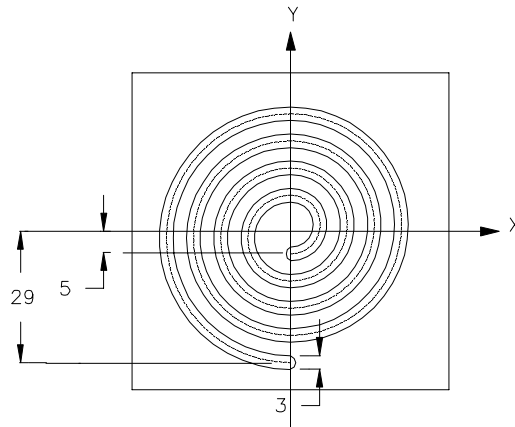
N14 Z20 C3600 C40=20 F125

Mill the helix. The rotary axis turns ten times (C3600)

N15 G0 Y100

Move tool away from the workpiece

Example 5. Spiral in the main plane (= facing plane) of a vertical milling machine.



The spiral has four turns and a pitch of 6 mm. It is produced with a rotary table (C-axis) rotating around the tool axis (Z-axis) and a simultaneous movement in the Y-axis. C40= 17

N10 G17

Define the main plane

N11 G54 T1 M6

Activate zero shift to the middle of the workpiece (G54).

Load tool T1 and its offsets. The tool is in the Z-axis (G17).

Mill diameter 3 mm.

N12 G0 X0 Y-5 Z2 C0 S3000 M3

Start the spindle. Move tool to start position.

N13 G1 Z-2 F75

Feed tool in Z-axis: 2 mm depth into the workpiece.

N14 Y-29 C1440 C40=17 F200

Mill the spiral (Rb=5, Re=29, therefore C40=17). Rotate the C-axis four times (C1440).

N15 G0 Z100

Move tool away from workpiece

5.3 G2/G3 Circular interpolation (CW/CCW)

To execute a Clockwise (G2) or Counter-Clockwise (G3) circular movement at a specified feedrate.

Format

Full circle.

G2/G3 [Centre point coordinates]

Arc less than or equal to 180°:

G2/G3 [Linear axis's end point coordinates] R...

Arc less or greater than 180°:

G2/G3 [Centre point coordinates] [Linear axis's end point coordinates]

G2/G3 [Centre point coordinates] B5=...

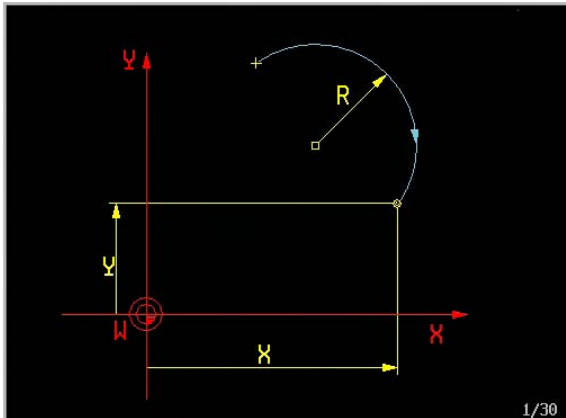
2.5D Interpolation

G2/G3 [Centre point coordinates] [Arc end point coordinates]
[Linear or rotary axis's end point coordinate]

Helix

G2/G3 [Centre point coordinates] [Arc end point coordinates]
[Linear or rotary axis's end point coordinates] [Pitch]

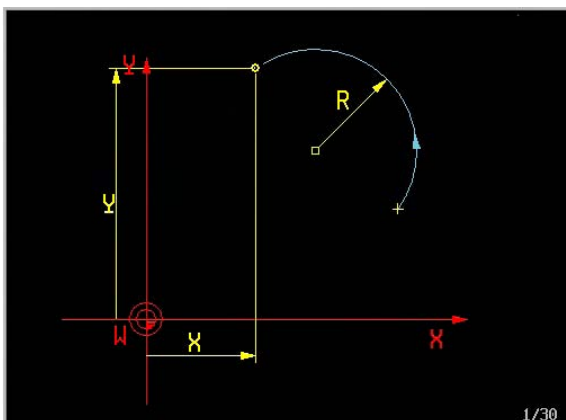
G2/G3 [Centre point coordinates] [Pitch] B5=...



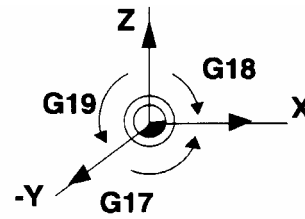
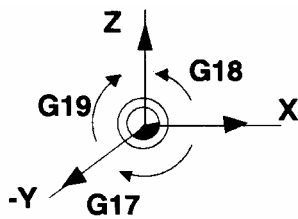
G2

G Circular counter clockwise
X Endpoint coordinate
Y Endpoint coordinate
Z Endpoint coordinate
B Endpoint angle
C Endpoint angle
I Center point in X / pitch in X
J Center point in Y / pitch in Y
K Center point in Z / pitch in Z
R Circle radius
B1= Angle
B2= Polar angle
B3= Polar angle for center
B5= Angle of arc
?90= End-/centrep. abs. (X,Y,Z..I,J,K)

?91= End-/centrep. incr.(X,Y,Z..I,J,K)
L1= Path length
L2= Polar length
L3= Polar length for center
P1= Point definition number



G3



The direction of circular movement is decided by looking in the negative direction of the tool axis towards the main plane.

End point coordinates

X, Y, Z	Endpoint coordinate
A, B, C	Endpoint angle
B1=	Angle
L1=	Path length
B2=	Polar angle
L2=	Polar length
P1=	Point definition number

Center point coordinates

I	Center point in X
J	Center point in Y
K	Center point Z / Pitch in Z
B3=	Polar angle for center
L3=	Polar length for center

Circle parameters

R	Circle radius
B5=	Angle of arc

Notes and usage

Modality

This function is modal with G0, G1, G6 und G9.

Modal words

F, F1=, F3=, F4=, F6=, S, S1=, T, T1=, T2=, M, M1=, H, Ennn, U, V, W
 D (Angle oriented spindle stop, only in combination with M19),
 I, J, K, R inside G182
 I, I1=, J1=, K1=, R, R1=, X1=, Y1=, Z1= inside ICP
 A40=, B40=, C40=, R for feed calculations rotary axes.

Circle in the main plane

Circular arc up to 180 degrees

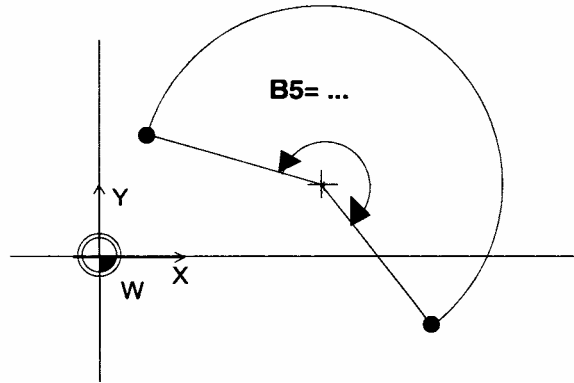
Using the end point's coordinates together with the arc radius or the coordinates of the arc's centre point programs an arc movement up to 180 degrees.

The arc radius is programmed with the R-word. This is a dimension word without a sign.

Circular arc greater than 180 degrees

An arc movement greater than 180 degrees can only be programmed with the coordinates of the end point and of the arc's centre point.

Angle of circular arc (B5=)



An arc of any angle between 0° and 360° can also be programmed with the centre point coordinates and the angle of the arc. The angle is programmed with the word B5= in decimal degrees and without sign.

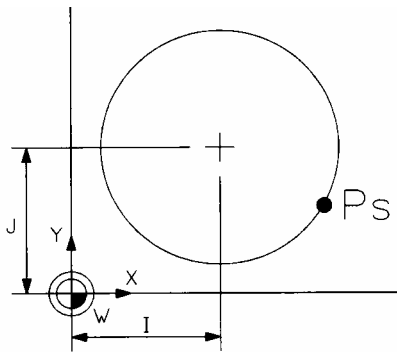
Programming a complete circle

After making a complete circle the tool is back at its start point. This movement is programmed with:

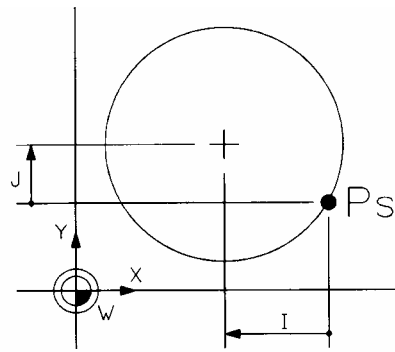
- the direction of movement on the circle,
- the coordinates of the centre point.

Note: If the end point coordinates are also programmed in the block, no circular movement will be executed.

Centre point coordinates



Absolute (G90) coordinates
Ps = Start point



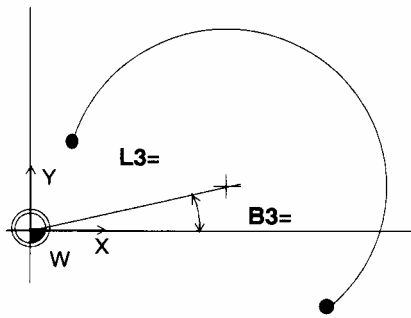
Incremental (G91) coordinates

Absolute centre point coordinates are related to the program zero point W.
Incremental centre point coordinates are measured from start point to centre point.

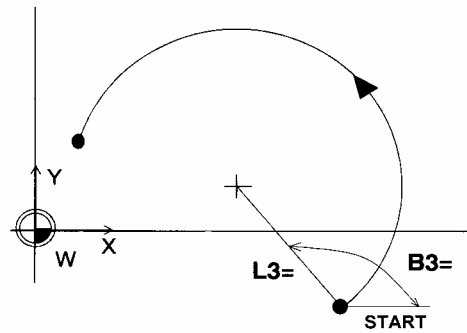
Centre point coordinates in different planes

N... G2/G3	I	(X-axis)	J	(Y-axis): XY-PLANE (G17)
N... G2/G3	I	(X-axis)	K	(Z-axis): XZ-PLANE (G18)
N... G2/G3	J	(Y-axis)	K	(Z-axis): YZ-PLANE (G19)

Polar centre point



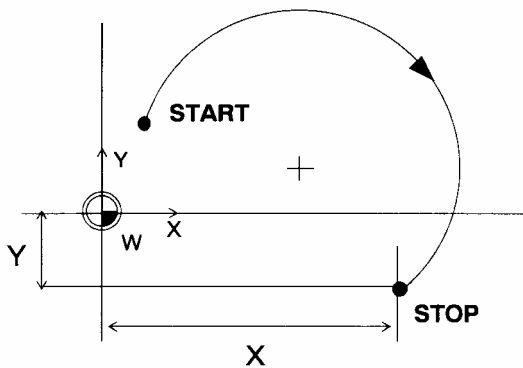
Absolute (G90) coordinates
L3=... B3=...



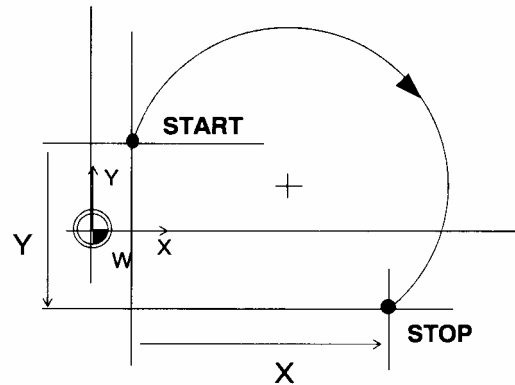
Incremental (G91) coordinates

The polar centre point coordinates are used in the plane defined by G17, G18 or G19.

Cartesian end point coordinates



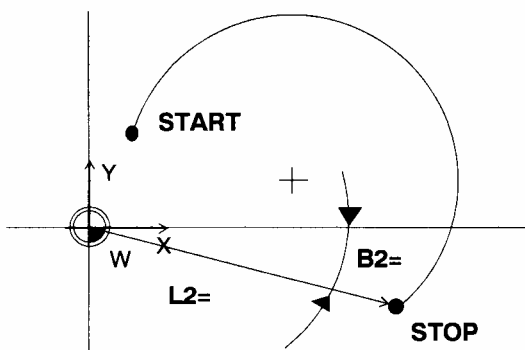
Absolute (G90) coordinates



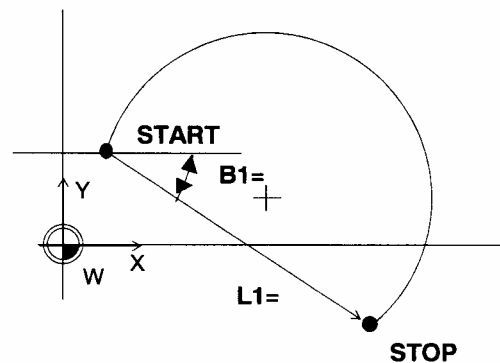
Incremental (G91) coordinates

(G17) X... Y... (G18) X... Z... (G19) Y... Z...

Polar end point coordinates and one coordinate and angle



B2=... L2=...



B1=... L1=...

These end point coordinates are used in the plane defined by G17, G18 or G19.

Defined point (G78)

A previously defined point (P-word) can be used to program the end point of a circular movement.

Feedrate in main plane

The programmed feedrate is the feed on the circle.

Radius compensation in main plane (G40 - G44)

Radius compensation on contours defined with linear and circular movements in the main plane is available. Refer to the functions G40, G41/G42 and G43/G44 for additional information.

A correction of the feedrate with circular movements and radius compensation depending on the shape of the contour and the radius of the mill is available. Refer to CONSTANT CUTTING FEED with the function G41/G42 for additional information.

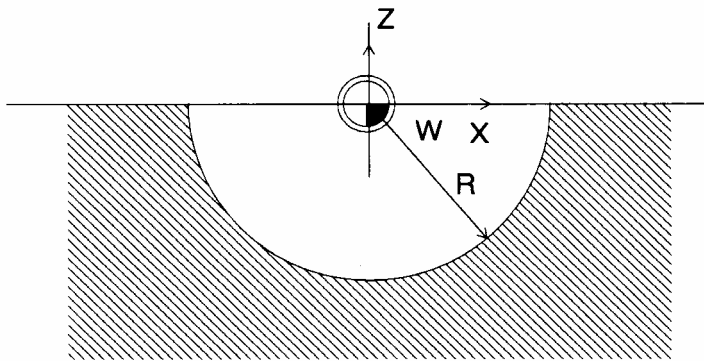
Axis rotation (G92/G93 B4=)

A circular movement in a rotated main plane can be used.

A circular movement not in the main plane

It is possible for a special tool such as a ball cutter, to be controlled so that it cuts in a direction being not parallel to the main plane. In these circumstances only Cartesian absolute or incremental dimensions can be used to program the end point and centre point coordinates.

Radius compensation is not available.



Circular movement in XZ-plane; tool in Z-axis

Circular arc up to 180°

An arc movement up to 180 degrees is programmed by using the end point's coordinates together with the arc radius (R-word) or the Cartesian coordinates of the arc's centre point.

Circular arc greater than 180°

An arc movement greater than 180 degrees can only be programmed with the Cartesian coordinates of the end point and the arc's centre point.

Addresses for end point and centre point

The addresses given in the table under the heading CENTRE POINT COORDINATES IN DIFFERENT PLANES also apply to arc movements, which are not in the current main plane. The addresses of the centre point define the plane in which the circle is to be milled.

Axis rotation (G92/G93 B4=)

A circular arc is not in the plane defined by the active G-function for plane selection (G17, G18 or G19) is not allowed in a plane of which one axis is rotated. E.g. if G17 is active and the XY-plane rotated 30°, the X-axis is rotated. So a circular movement in the XZ-plane is not allowed. An error message is displayed to this effect.

Linear axes U, V and W

If a machine tool is equipped with linear axes parallel to the main axes, circular movements can be used with these axes.

Polar coordinates or radius compensation cannot be used.

Programming a circular arc

For an arc movement up to 180 degrees, either the radius (R-word) or the Cartesian centre point coordinates (absolute or incremental) can be programmed.

For an arc movement greater than 180 degrees only Cartesian coordinates can be used for programming the centre point.

Programming end point and centre point

Both coordinates of the endpoint must be programmed because they determine the plane in which the circular movement occurs.

Centre point coordinates are defined by: I for the U-axis; J for the V-axis; K for the W-axis. The tables below indicate which addresses are used with different planes.

Main axis X and a linear axis

	XV-plane	XW-plane
End point	X and V	X and W
Centre point	I and J	I and K

Main axis Y and a linear axis

	YU-plane	YW-plane
End point	Y and U	Y and W
Centre point	J and I	J and K

Main axis Z and a linear axis

	ZU-plane	ZV-plane
End point	Z and X	Z and Y
Centre point	K and I	K and J

Combination of linear axes

	UV-plane	UW-plane	VW-plane
End point	U and V	U and W	V and W
Centre point	I and J	I and K	J and K

A circular movement with a simultaneous movement in a third axis

The CNC control can use a special interpolation procedure (2.5D), to coordinate a circular movement in a main plane and a third axis' movement, so that a tool travels the correct paths from the start point to the end point.

Circle in the main plane

The normal programming methods for a circle in the main plane defined by G17, G18 or G19 are used.

Radius compensation with the circular movement can be used. The tool must tangentially enter and leave the workpiece.

If the third axis is the tool axis, using one of the addresses given in the table below programs it.

	G17	G18	G19
Plane	XY-plane	XZ-plane	YZ-plane
Tool axis	Z	Y	X

Circle not in the main plane

When the circular movement is not in the main plane, the rules given in A CIRCULAR MOVEMENT NOT IN THE MAIN PLANE must be used for programming the movement.

The Cartesian coordinates of the centre point define the plane. An arc radius (R-word) cannot be used.

The table below lists the addresses, which are used for different planes.

	XY-plane	XZ-plane	YZ-plane
End point	X and Y	X and Z	Y and Z
Centre point	I and J	I and K	J and K
Tool axis	Z	Y	X

Third axis is a rotary axis

If the circular movement is executed in the main plane or in another plane as well, the third axis is not restricted to the tool axis, but a rotary axis programmed with the address A or B can also be used. In this case a simultaneous movement of the linear axes performing the circular movement in the defined plane, and the rotary axis occur.

Helix interpolation

Programming the following can mill a helix on any cylinder surface:

- circular movement in the main plane as described
- the pitch of the helix
- (if necessary) the end point of the linear movement.

	G17	G18	G19
Tool axis	Z	Y	X
Centre point	I and J	I and K	J and K
	or	or	or
	B3= and L3=	B3= and L3=	B3= and L3=
Angle of arc	B5=	B5=	B5=
Pitch of helix	K	J	I

Programming the arc angle

The value of 'B5=' can be from 0 to 999999 degrees, which is approximately 900 revolutions.

Programming the tool axis

A helix movement can also be programmed by using the addresses given in the table below.

	G17	G18	G19
Tool axis	Z	Y	X
Circle end point	X and Y	X and Z	Y and Z
Centre point	I and J	I and K	J and K
Pitch of helix	K	J	I

When these alternative addresses are used, the movements have to be programmed so that the circular movement and the tool axis movement reach their end positions at the same time.

Third axis is a rotary axis

The third axis is not restricted to the tool axis, but a rotary axis programmed with the address A or B can also be used. In this case a simultaneous movement of the linear axes performing the circular movement, and the rotary axis occur.

Radius compensation with helix interpolation

Radius compensation with the circular movement can be used during helix interpolation. The tool must tangentially enter and leave the workpiece.

General remarks with G2/G3

Cancellation

The function G2 or G3 is cancelled by any other function of group A, or at end of program (M30), or by CLEAR CONTROL.

Checking of centre point coordinates

When centre point coordinates are used, the radius of the circular movement at the start is compared with the radius at the end. If the difference between the two values is greater than a Machine Constant setting, the CNC generates an error message and stops program execution.

Start next movement (G28 I3=..)

In general feed movements are executed without a stop between the blocks. This results in rounded corners.

G28 and parameter I3= allows to program if the next movement starts after a full stop of the tool or without a stop between the movements.

Refer to the function G28 for details.

Feed limitation (G28 I6=..)

With feed limitation the programmed feedrate is reduced to keep the axes following error to an acceptable minimum and therefore to improve the machining accuracy. Refer to the function G28 I6= for details.

Oriented spindle stop (D.. M19)

The D-word for the offset angle with oriented spindle stop must be programmed together with the function M19. Refer to the function M19 for details.

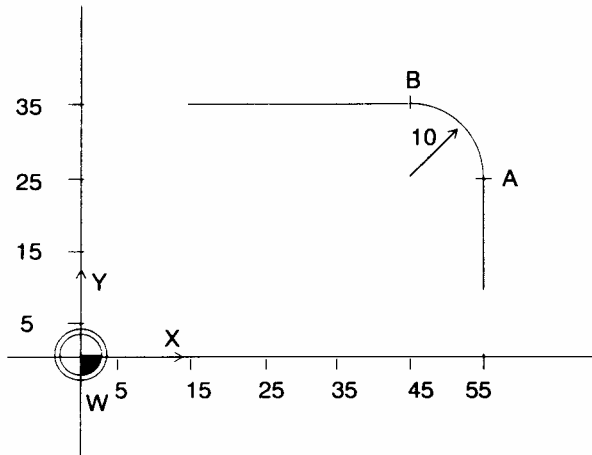
See also G303 M19 D..

Note: If a circle is programmed and one of its endpoints is within some microns from the startpoint, then at high feed values the cnc will not perform a circle but a linear movement directly to the endpoint.

Example: if difference between start and end position is 5 micron, then if the feed exceeds 1 m/min then a linear movement will be executed although G2/G3 has been programmed.

Examples

Example 1. Programming an arc radius



N10 G1 X55 Y25 F100

(A)

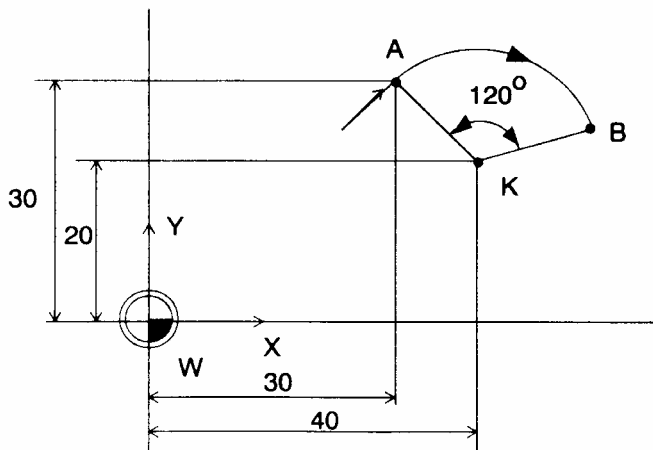
Move tool at set feedrate to the starting point A of the arc.

N20 G3 X45 Y35 R10

(B)

Move tool in a counter-clockwise direction (G3) to end point B

Example 2. Programming an arc angle.



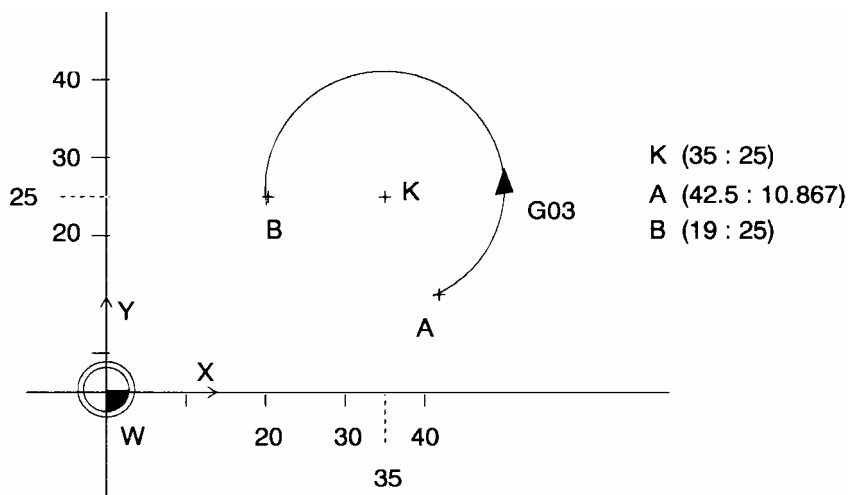
N10 G1 X30 Y30 F500

(A) Move tool at set feedrate to the starting point A

N11 G2 I40 J20 B5=120

(B) Move tool in a clockwise direction (G2) to end point: the angle of the arc is stated by B5=.

Example 3. Programming an arc > 180 degrees (Cartesian)



Absolute coordinates

N10 G1 X42.5 Y10.867 F200 (A)
N11 G3 X19 Y25 I35 J25 (B)

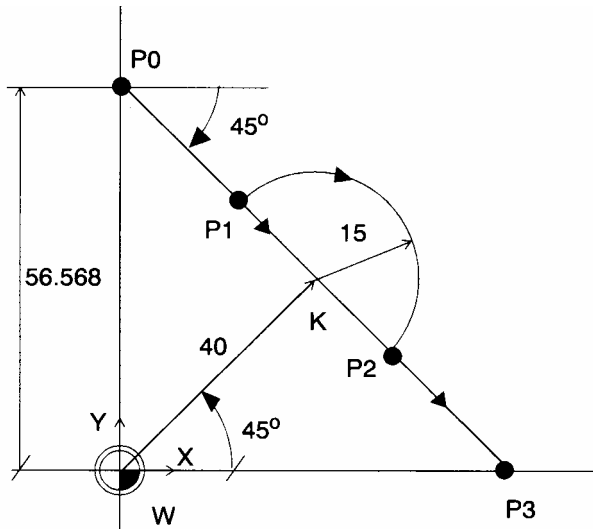
Move tool at given feedrate to the starting point A of the arc.
Move tool in a counter-clockwise (G3) direction to end point B. Centre point coordinates are stated by I and J.
Both the coordinates X and Y as well as I and J are absolute values with regard to the program zero point W.

Incremental coordinates

N10 G1 X42.5 Y10.867 F200 (A)
N11 G91
N12 G3 X-23.5 Y14.133 I-7.5 J14.133 (B)

Move tool at given feedrate to the starting point A of the arc.
Activate incremental coordinate mode (G91).
(B) Move tool in a counter-clockwise (G3) direction to end point B. Coordinates X and Y are increments from point A to B. The coordinates I and J are incremental values from A to the centre.

Example 4. Programming an arc with polar coordinates



N10 G0 X0 Y56.568 (P0)

Move tool rapidly to point P0.

N11 G1 B1=-45 L1=25 F200 (P1)

Move tool at programmed feedrate from P0 and P1. Incremental polar coordinates are used.

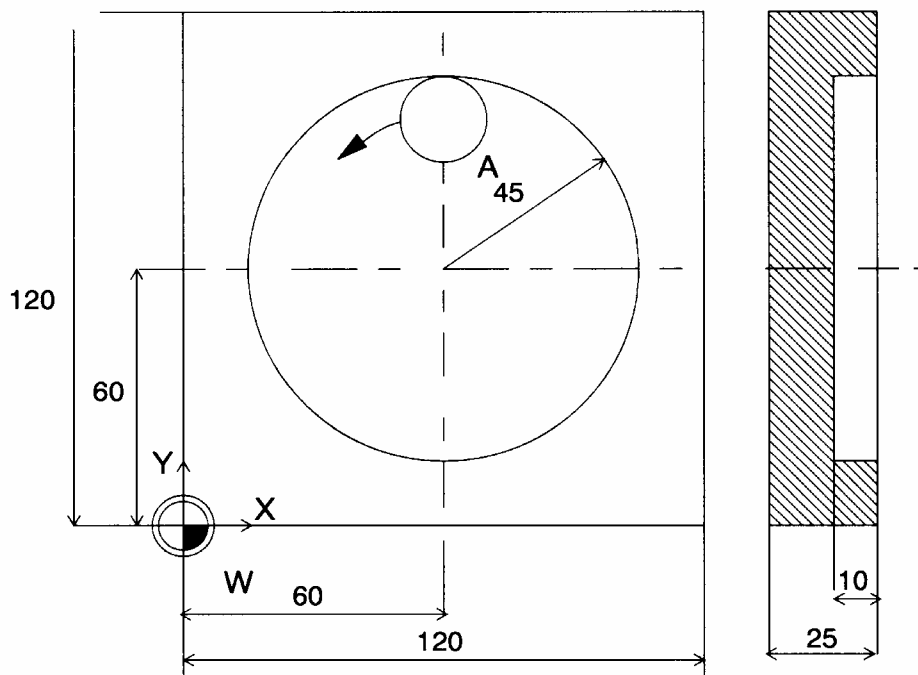
N12 G2 B1=-45 L1=30 B3=45 L3=40

(P2) Move tool in a clockwise (G2) direction from P1 to P2. For P2, incremental coordinates are used. The centre point is programmed with absolute polar coordinates (B3=, L3=).

N13 G1 B1=-45 L1=25 (P3)

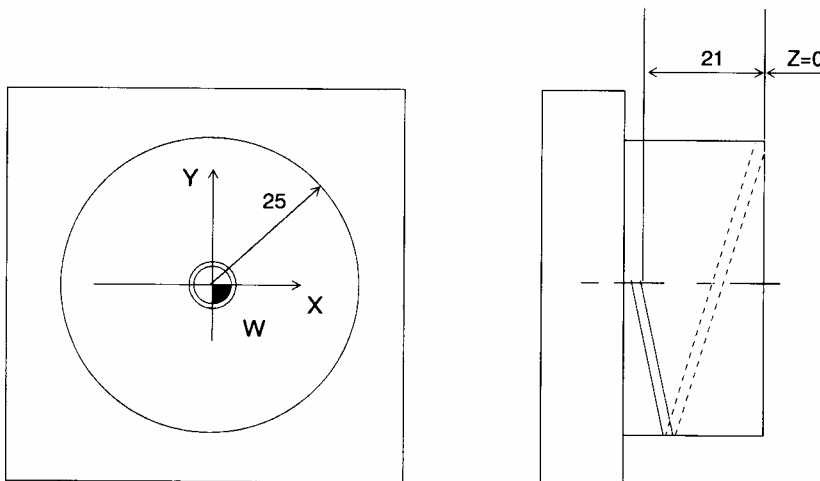
Move tool at set feedrate from P2 to P3.

Example 5. Programming a complete circular movement.



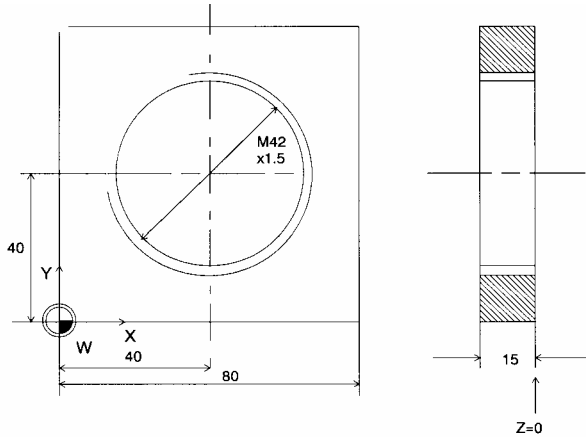
N9 G17 T1 M6	Activate XY-plane. Load tool T1 and its offset. (Mill diameter 16 mm)
N10 G0 X60 Y60 Z10 S315 M3	Start the spindle and move tool rapidly to the centre of the pocket (Z10)(where the tool is to enter the hole). The feedrate is 36 mm/min
N11 G1 Z-10 F36	Set the feedrate to 65 mm/min and feed tool depth.
N12 G43 Y105 F65	Move tool with feed to the wall (G43). To point A
N13 G41	Set radius compensation for a tool moving on the left handside (G41).
N14 G3 I60 J60	Mill the complete circle in a counter clockwise direction (G3).
N15 G40	Cancel radius compensation (G40).
N16 G1 Y90	Move tool away from milled surface of workpiece.
N17 Z100 M30	Retract tool out of workpiece.

Example 6. Programming a circular movement together with a simultaneous linear axis movement (2.5D Interpolation).



N10 G17 T1 M6	Activate XY-plane. Select tool 1 (diameter 3 mm) and its offsets.
N11 G0 X0 Y35 Z0 S1000 M3	Start the spindle and move the tool to the workpiece at 3000 rev/min
N12 G43	Activate radius compensation to the endpoint
N13 G1 Y25 F80	Move tool to the workpiece contour. Set linear feedrate to 80 mm/min
N14 G41 F120	Set radius compensation LEFT.
N15 G2 X-25 Y0 Z-21 I0 J0	Perform the simultaneous movement of the circle and third axis.
N16 G40	Cancel radius compensation.
N17 G1 X-35	Move tool away from workpiece.

Example 7. Programming a helix.



N10 G17
N11 T1 M6

N12 G0 X40 Y40 Z1.5 S400 M3
N13 G1
N14 G43 Y61 F120
N15 G42
N16 G2 I40 J40 K1.5 B5=4320

N17 G40
N18 G1 Y40
N19 G0 Z100

Define the main plane
Load tool 1 and its offsets. Spindle rotation 400 rev/min
(Thread mill cutter)
Start the spindle and move tool to starting position.
Set linear feed movement
Move tool with feed to (G43) the part.
Set radius compensation RIGHT (G42).
Mill the helix.
Programmed arc:
- circle centre (I and J)
- angle of the arc (B5=) 12 turns of 360 degrees
- pitch of thread (K).
Cancel radius compensation (G40).
Move tool away from the wall.
Retract the tool.

By using an alternative set of addresses, block 16 could be re-written as:
N16 G2 X40 Y61 Z-16.5 I40 J40 K1.5

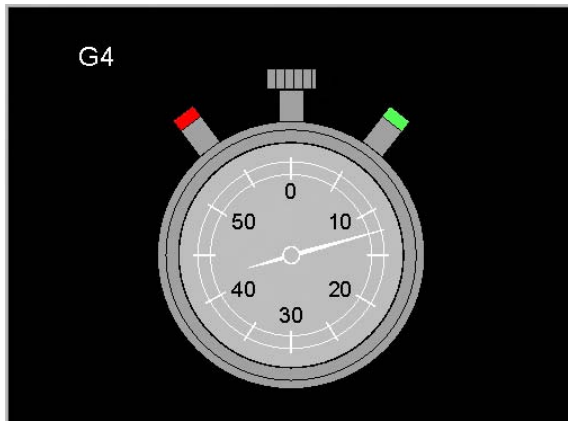
Programmed arc:
- circle end point (X and Y)
- depth (Z)
- circle centre (I and J)
- pitch of thread (K).

5.4 G4 Dwell time

During the execution of a program inserting a dwell period (time or number of revolutions).

Format

G4 X.. or D.. or D1=..



```
G   Dwell time
X   Dwell time in sec.
D   Dwelltime in revolutions of S
D1= Dwelltime in revolutions of S1
```

Notes and usage

Input values

Dwell period (D):	0,1 - 900 Seconds (15 Minutes).
Revolutions (D1=):	0 - 9.9

Example

N50 G4 X2.5

The above block causes a dwell of 2.5 seconds between two operations

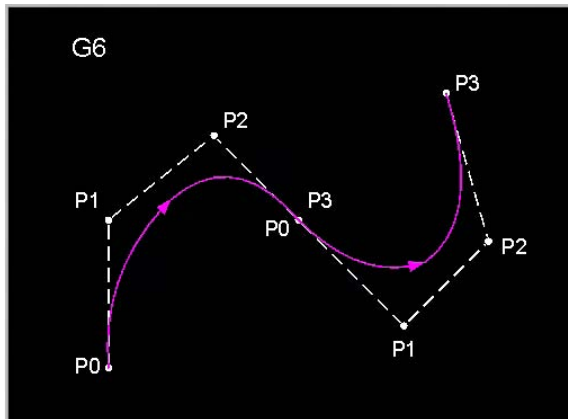
N60 G4 D2

The above block causes a dwell of 2 revolutions of the spindle between two operations

5.5 G6 Spline-interpolation

The spline interpolation of the CNC enables the partprogrammer to input a series of points and have the control fit a smoothly faired curve through them.

By using this function, machine dynamic response is improved and leads to smoother tool movements and improved machining accuracy.



```
G   Spline interpolation
X   Endpoint (X-axis)
Y   Endpoint (Y-axis)
Z   Endpoint (Z-axis)
X51= First spline coefficient
Y51= First spline coefficient
Z51= First spline coefficient
X52= Second spline coefficient
Y52= Second spline coefficient
Z52= Second spline coefficient
X53= Third spline coefficient
Y53= Third spline coefficient
Z53= Third spline coefficient
X61= First support point (X-axis)
Y61= First support point (Y-axis)
```

```
Z61= First support point (Z-axis)
X62= Second support point (X-axis)
Y62= Second support point (Y-axis)
Z62= Second support point (Z-axis)
```

Bezier Splines

X, Y, Z

X61=, Y61=, Z61=

X62=, Y62=, Z62=

Endpoint (Z-axis)

First support point (Y-axis)

Second support point (Y-axis)

Cubic Splines

X51=, Y51=, Z51=

X52=, Y52=, Z52=

X53=, Y53=, Z53=

First spline coefficient

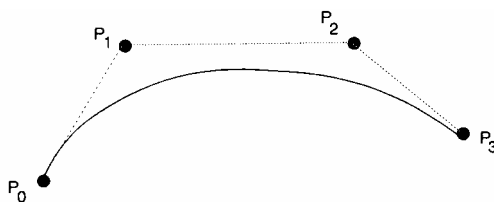
Second spline coefficient

Third spline coefficient

Formats with Bezier splines

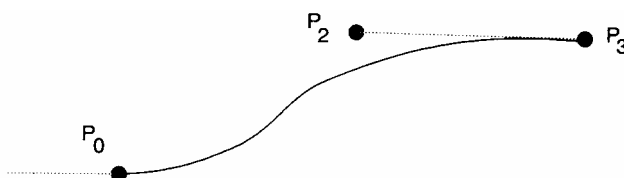
Spline with three vertex points

G6 X61=.. Y61=.. Z61=.. X62=.. Y62=.. Z62=.. X.. Y.. Z..

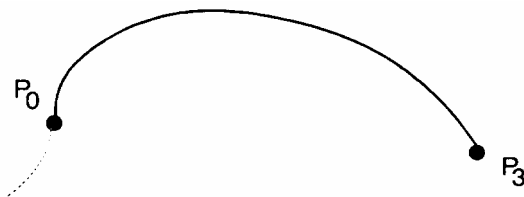


Spline with two vertex points and constant tangent with previous spline

G6 X62=.. Y62=.. Z62=.. X.. Y.. Z..



Spline with constant curvature with previous spline
G6 X.. Y.. Z..



Formats with cubic splines

Spline with all coefficients defined

G6 X51=.. Y51=.. Z51=.. X52=.. Y52=.. Z52=.. X53=.. Y53=.. Z53=..

Spline with constant tangent with previous spline

G6 X52=.. Y52=.. Z52=.. X53=.. Y53=.. Z53=..

Spline with constant curvature with previous spline

G6 X53=.. Y53=.. Z53=..

Notes and usage

Modality

This function is modal with G0, G1, G2, G3 und G9.

Machine constants

For using splines MC262 and MC265 must be greater then 0.

Notes and usage with Bezier splines

Definition bezier splines

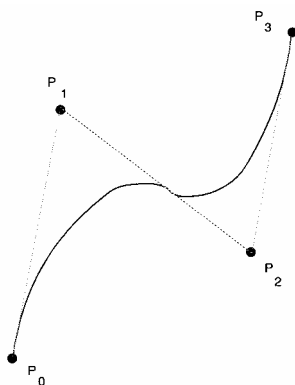
A Bezier spline is a spline defined by four points, the so-called vertex points.

The first point of the spline is the end point of the previous movement. The curve passes through this point. The first and second vertex points control the shape of the spline. The curve does not pass through these points. The last point is the end point of the spline and this is, like the first point, a point through which the curve passes.

Coordinates of the vertex points

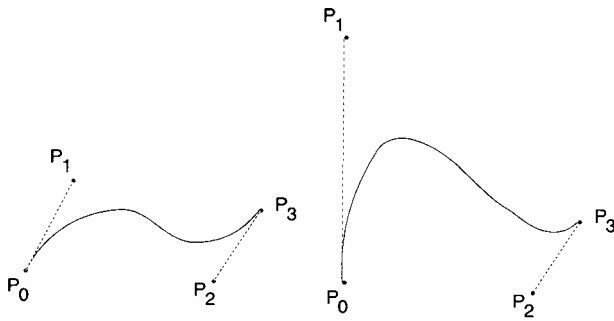
Only absolute Cartesian coordinates of the main axes X, Y and Z can be used for programming the vertex points. The coordinates are related to the program zero point. The coordinates of all three axes must be entered, even if not changed.

Tangent at start of spline



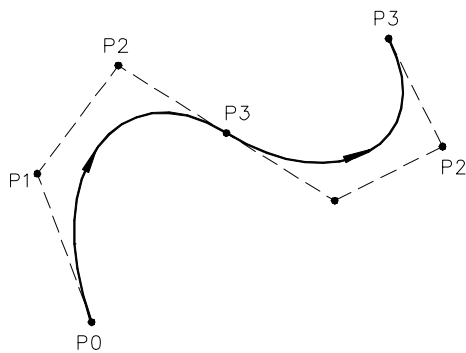
From the coordinates of the first vertex point (X61=, Y61=, Z61=) the tangent at the start of the spline is calculated as the line through the end point of the previous spline and the first vertex point.

Changing the location of the first vertex point



In the illustration three points of the spline are fixed and just the location of the first vertex point is changed. Notice that the shape of the curve changes.

Tangential continuity at start of spline

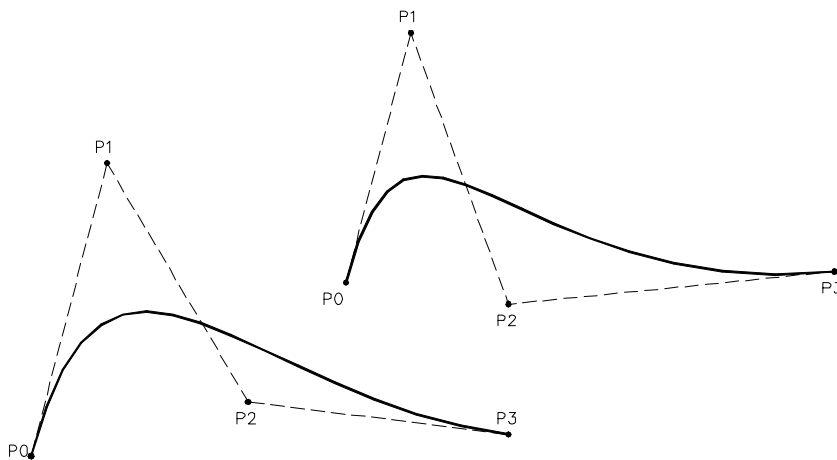


If the first vertex point is not programmed, the tangent at the end of the previous spline is used as the tangent at the start of the spline.

Tangent at end of spline

From the coordinates of the second vertex point (X62=, Y62=, Z62=) the tangent at the end of the spline is calculated as the line through the second vertex point and the end point.

Changing the location of the second vertex point



In the illustration three points of the spline are fixed and just the location of the second vertex point is changed. Notice that the shape of the curve is influenced by this change in location.

Constant curvature

If the first and second vertex points are not programmed, constant curvature between the splines is assumed.

Getting started with a group of bezier splines

If a curve or surface is defined by a group of G6-blocks with Bezier splines, the first spline must be programmed with three vertex points.

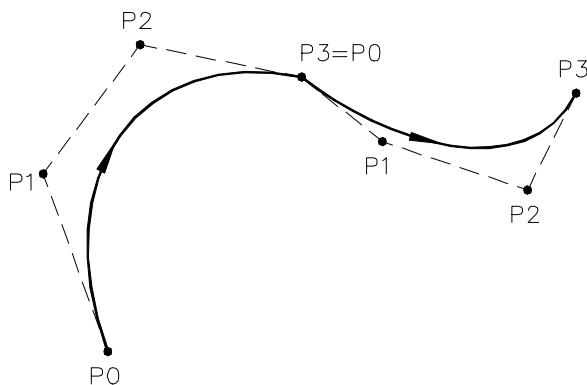
One way of doing this is to program a spline with three equal vertex points. These points are on a straight line, which is tangent to the spline in the end point of the "line".

Example: N10 G1 X10 Y10 Z10
 N11 G6 X61=25 Y61=25 Z61=10 X62=25 Y62=25 Z62=10 X25 Y25 Z10
 The block N11 generates a straight line in the XY-plane.

Connecting splines

The three different types of Bezier splines can be connected to form a curve or surface.

Corner between curves



If a corner between the splines, the second spline must be programmed with three vertex points.

Radius compensation

Radius compensation on splines is not available, so the toolpath must be programmed when using splines.

Connection between a line or circle and a bezier spline

If the previous movement is linear or circular, its end point can be used as the start point of a Bezier spline with three vertex points.

Because radius compensation with splines is not available, the end point of the line or circle must also be programmed without radius compensation.

Absolute and incremental programming (G90/G91)

If G91 is active during a section with G6-blocks, the incremental programming is ignored with the G6-blocks and executed with the linear or circular movements.

It is advised to use always absolute programming (G90 active), when splines are involved.

Notes and usage with cubic splines

In-depth knowledge of the cubic spline coefficients is essential. Calculations are usually done by a CAD system. The toolpath is then also generated.

Polynomial expression for cubic spline

The cubic spline is defined by a polynomial expression.

For e.g. the X-axis, the polynomial expression for the cubic spline is (t is a parameter with $0 < t < 1$):

$$X = [X53]t^3 + [X52]t^2 + [X51]t$$

For the other axes the same polynomial is used with the programmed coefficients and the same parameter t for all axes.

These three polynomial expressions define a patch of the surface in space.

Determining the coefficients

It is not easy to calculate the coefficients of the cubic polynomial. A good knowledge of spline coefficients is required. These calculations are mostly performed by a CAD-system, which also generates the toolpath.

Tangential continuity at start of spline

If the first order coefficients are not programmed, the tangent at the end of the previous spline is used as the tangent at the start of the spline. From this tangent line the missing coefficients are calculated.

Constant curvature

If the first and second order coefficients are not programmed, constant curvature between the splines is assumed. The missing coefficients are calculated by the control.

Note: Omitting coefficients is only possible if the missing coefficients can be calculated from previous spline blocks

Getting started with a group of cubic splines

If a curve or surface is defined by a group of G6-blocks with cubic splines, the first spline must be programmed with all three coefficients.

Notes and usage with both types of splines**Mixing bezier and cubic splines**

Both types of splines can be mixed at will.

Graphic simulation

The splines can be displayed in synchron graphics. Other graphic modes ignore the spline function.

Cancellation

The function G6 is cancelled by one of the functions G0, G1, G2 or G3 at end of program (M30), CLEAR CONTROL or the softkey CANCEL PROGRAM.

Plane selection (G17/G18/G19)

Splines are independent of the selected plane. The points (coefficients) determine the plane in which the spline is made.

Zero point shift (G92/G93)

If a zero point shift is programmed between G6-blocks, it is ignored with the G6-blocks and used with the other blocks.

Axis rotation (G92/G93 B4=..)

Axis rotation is ignored with the G6-blocks and used with the other blocks.

Scaling and mirror image (G73)

Scaling and mirror image are ignored with the G6-blocks and used with the other blocks.

Start next movement

In general feed movements are executed without a stop between the blocks. This results in rounded corners.

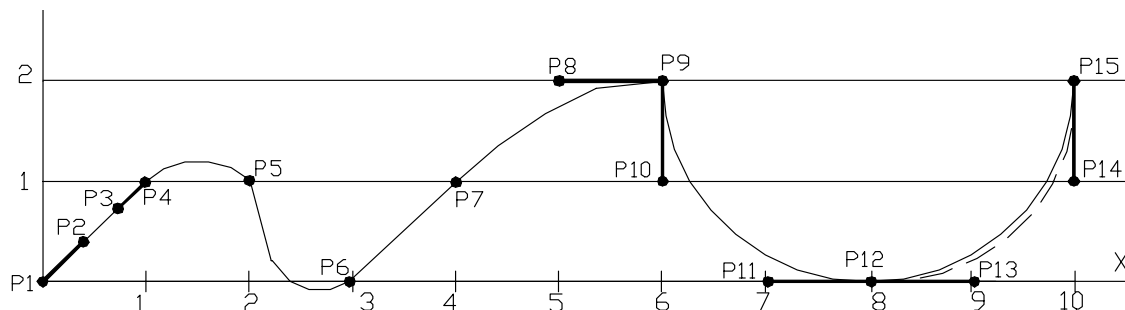
G28 and parameter I3= allows to program if the next movement starts after a full stop of the tool or without a stop between the movements.

Refer to the function G28 for details.

Restrictions

1. Corner accuracy (G28 I3=2 or 3) and feed limitations (G28 I6=) cannot be used together with spline functions
2. The BTR-function cannot be used with splines.
3. When geometric calculation is active (G64) the spline-interpolation cannot be used.

Example:



N17001 (Spline curve)

N1 G98 X2 Y-6 Z-2 I10 J10 K10

N2 G17

N101 G0 X0 Y0 Z0 F500

Approaching starting position of curve (P1)

N102 G6 X1 X61=0.3 X62=0.7 Y1 Y61=0.3 Y62=0.7 Z0.001 Z61=0 Z62=0 First curve element. Straight line. Touches P1-P2 and P3-P4. End point is P4. All coordinates should be entered. Select a straight line.

N103 X2 Y1.001 Z0

Curve passes through P5

N104 X3 Y0 Z0.001

Curve passes through P6

N105 X4 Y1 Z0

Curve passes through P7. More points should be added if the curve differs from the required shape.

N106 X6 X62=5.7 Y2 Y62=2 Z0.001 Z62=0 Curve passes through P9 and touches line P8-P9.

N107 X8 X61=6 X62=7.5 Y0 Y61=1.5 Y62=0 Z0 Z61=0 Z62=0.001 New curve with sharp transition is defined. First curve element starts in P9 and touches P9-P10 and P11-P12. End point is P12.

N108 X10 X61=8.5 X62=10 Y2 Y61=0 Y62=1.5 Z0.001 Z61=0.001 Z62=0 New curve with tangential transition is defined. First curve element starts in P12 and touches P12-P13 and P14-P15. End point is P15. The radius of curvature may be adjusted in P15 by changing distance P14-P15.

N109 G0 X0 Y0 Z0

Return to starting position.

N110 M30

Note: In G6, the same coordinates should be different in two blocks (Z0 and Z0.001)

5.6 G7 Tilting working plane

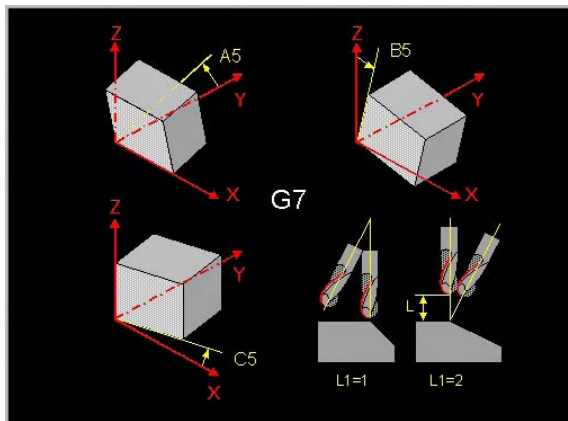
Programming of a tilt-operating plane for four or five axis machines.

The position of the operating plane can be tilted using the function "Tilt operating plane". The operation, which has then been programmed in the principal plane, (G17, G18) can then be implemented within the tilt-operating plane. The tool axis is then orientated vertically in the new plane.

The tilt of the operating planes is defined and implemented using the G7 function.
Refer to chapter "Tilting of the operating plane".

Format

G7 {A5=.. or A6=..} {B5=.. or B6=..} {C5=.. or C6=..} {A7=..} {B7=..} {C7=..} {B47=..} {L1=..} {L2=} {L..}



G Tilting working plane
L Tool length offset
B47= E-par. for rotation mainplane
A5= Angle of rotation absolute
B5= Angle of rotation absolute
C5= Angle of rotation absolute
A6= Angle of rotation incremental
B6= Angle of rotation incremental
C6= Angle of rotation incremental
B7= E-par. for position in B
C7= E-par. for position in C
L1= 0=No move., 1=rot.axes, 2=tooltip
L2= -/+1,2,3 = Neg/Pos A,B,C angle

Notes and usage

Modality

This function is modal and stay active until another G7 is programmed.

G-functions, which are not permitted, if G7 is switched on

IF G7 is switched on, the following (modal) G-functions are not allowed to be active:

G6, G9, G19, G33, G41, G42, G43, G44, G61, G64, G73, G141, G182, G197, G198, G199, G200, G201, G203, G204, G205, G206, G207, G208

When G7 is turned on, the following (modal) G-functions with the addresses below must not be active:
G54 I1 B4=... and G93 B4=...

G functions within G7, which are not permitted

If G7 is active, the following G functions are not permitted:

G6, G19, G66, G67, G182, G339

G functions, which are not permitted, if G7 is switched off

If G7 is switched off, the following (modal) G functions are not allowed to be active:

G9, G41, G42, G43, G44, G61, G64, G73, G141, G197, G198, G199, G200, G201, G203, G204, G205, G206, G207, G208

When G7 is turned off, the following (modal) G-functions with the addresses below must not be active:
G54 I1 B4=... and G93 B4=...

G7 function

The freely programmable operating plane is defined using the new G7 function:

The new plane becomes active with the original null point.

The tool is orientated vertically in the new plane. The axes, which move depends upon the machine configuration and the programming.

The display shows the coordinates in the new (tilt) plane.

The manual operation is orientated in accordance with the new plane.

Space angle

A5=, B5=, C5= Defines the absolute angle, by which the operating plane is rotated about the corresponding positive axis.

A6=, B6=, C6= Defines the incremental angle, by which the operating plane is rotated about the corresponding positive axis.

Value falls between -359.999 and 359.999 [degrees]

Definition of new operating planes

Tilting of the operational plane can be defined in two ways:

- Programming with A5=, B5= or C5= parameters. In this way, the absolute tilts about the corresponding positive axes are defined. The tilts are implemented as follows:
 1. the active G7 tilt is raised
 2. C5= tilt about the machine fixed positive Z axis
 3. B5= tilt about the positive Y axis
 4. A5= tilt about the positive X axis
- Programming with A6=, B6= or C6= parameters. The incremental tilts about the current corresponding positive axes are defined in this way. The tilts are implemented as follows:
 1. C6= tilt about the current G7 positive Z axis
 2. B6= tilt about the current G7 positive Y axis
 3. A6= tilt about the current G7 positive X axis

The programming is independent of the machine configuration. The plane tilt is calculated with reference to the current null point. The movement is dependent upon the machine configuration.

Query a calculated angle position

A7=, B7=, C7= Holds the number of the E-Parameters, in which the computed angle of the corresponding rotary axis is set.

B47= Contains the number of the E-Parameter, in which the computed angle of the main plane is set.

Alternative tilting possibilities within moving range of the machine

The CNC checks, which tilting possibilities within the moving range of the rotary axes are possible (to the left or to the right).

- No tilting possibilities, then error message is given (P307)
- By only one tilting possibility this will be executed.
- By two tilting possibilities, those with the shortest movement will be executed (L2=0 or not programmed). The shortest movement is not always possible.

With the address L2= can be controlled, which tilting possibilities must be executed. By L2=1/2/3 the A/B/C-axis is positioning so, that a positive angle will be reached. By negative L2= a negative angle will be reached.

Tool vertical on the defined tilt plane

The G7 tilt movement takes place interpolating with the power traverse. It tilts the tool axis to the defined plane. The axes, which are moved, depend upon the type of movement L1=:

- L1=0 the rotary axes do not move (start position).

Comment: The tilt movement can then be implemented, using the E parameters loaded into A7=, B7= or C7=. This movement must then be programmed manually.

- L1=1 Interpolate only the rotary axes, which do not move the linear axes.
- L1=2 Interpolate the rotary axes and to that end execute a "compensatory movement". In this way the tooltip remains in the same position with respect to the workpiece.

Tool length allowance

If the tilting motion takes place about the tooltip (L1=2), I defines an allowance in the tool direction between the programmed endpoint and the tooltip.

Switching off the G7 function

The operation of G7 remains active until G7 is switched off. G7 is switched off by the programming of G7 without parameters or by G7 L1=1 positioning of the rotary axes on the workpiece null point.

G7 is not switched off by M30 or <Program interrupt>. After switching on the control G7 is permanently active. Travel in the G7 plane is then possible. G7 is switched off in accordance with reference point travel or <Reset CNC>.

Note: It is recommended that, at the start of every G7 program, that a G7 without parameters is programmed. In this way, during the start-up of the program (interrupt within the tilted plane and the new start), the plane is always reset. Without this G7 at the start, the first part of the program will be implemented in the tilted plane rather than in the untilted plane. This programming is similar to programming with G17/G18 - different null points or different tools.

Rotary axes

Rotary axes can be programmed in the tilted planes in the normal way. It is the programmer's responsibility to ensure compatibility of the rotary axes with the G7 tilt.

Absolute position G74

If G7 is active, the "Absolute position" G74 is referred to the machine coordinates. This is the same as in V3.3x.

Graphics

The graphics display the G7 plane as the main picture. The screen is refreshed whenever G7 becomes active.

If G7 is active, the position between tool and workpiece is displayed.

Display

If G7 is active, a yellow icon is displayed on the screen behind the tool number. By means of a small "p" on the right next to the "axes characters", an indication is given as to whether the display relates to the tilted operating planes or to the machine coordinates. The operating status has been enhanced with the current reading of the programmed G7 space angle.

A new soft key (Jog to the G7 plane) appears in the "Jog operation type" soft key group. This soft key is used to switch between the tilted operating planes and the machine coordinates. If the position is displayed in machine coordinates, the actual position of the tooltip is shown.

Change of tool

If G7 is active, a tool change is, depending of the IPLC program, yes or no permitted (fault report).

If a tool change is not permitted, G7 must first be deselected. G7 must then be selected again, in order to resume work in the tilted plane following the tool change.

Example:

N100 G7 B5=45 L1=1	(plane is set)
N110 T14	(tool preselect)
..	
N200 G0 Z200	(the tool axis is withdrawn)
N210 G7 B5=0 L1=1	(deselect G7)
N220 M6 T14	(tool change)
N230 G0 X.. Y.. Z..	(power traverse to the new start position)
N240 G7 L1=1 B5=45	(face is rotated again in the G7 plane)

Palette, tilt face or tool change

While G7 is active, is depending of the IPLC program, a change of palette, tilt face or tool cannot be implemented. A fault is displayed and the program must be interrupted. Before such a change, G7 must be deactivated.

Tilt operating plane with M53/M54

During mixed operation with G7 and M53/M54, the tilt face positioning M53/M54 with M55 must be relinquished before programming G7. In this way, the face offset, which is active under these conditions, is relinquished.

M functions, which are not permitted if G7 is switched on

If G7 is switched on, the following M functions are not permitted to be active:
M53, M54

M functions, which are not permitted within G7

If G7 is active, the following M functions are not permitted:
M6, M46, M53, M54, M60, M61, M62, M63, M66

Fault reports

- | | |
|------|--|
| P48 | G-function not allowed after rotation
The programmed combination is not allowed. (G54 I1 B4=... and G93 B4=...) |
| P77 | G-function and Gxxx not permitted
The combination of G-functions is not permitted. For example: Is G7 programmed and G41 is active, an error message P77 "G-function and G41 not permitted" is given. |
| P306 | Plane not clearly defined
The G7 plane is defined with a mix of absolute (A5=, B5=, C5=) and incremental (A6=, B6=, C6=) angles.
Resolution: Use only absolute or incremental angles. If required, several G7 definitions with incremental angles behind one another can be defined. |
| P307 | Programmed plane not attainable
The defined G7 tilt position, owing to the restricted range of the rotary axis, cannot be attained. |

Machine constants

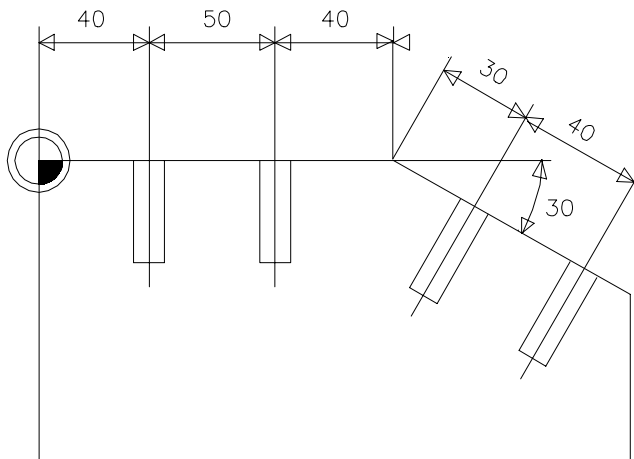
MC312 free operating plane (0=off, 1=on)

Activates the free operating plane. The G7 function can be programmed.

MC755 free operating plane: rotation (0=coordinates cross, 1=axes)

If the desired rotation of the operating plane is compatible with the rotation of a rotary axis, an adjustment may be made to determine whether the relevant rotary axis or the coordinates cross is rotated.

e.g. on a machine with (real C axis) the programming G7 C5=30 and MC755=0 generates a rotation of the coordinates cross through -30° and, if MC755=1, a rotation of the C axis through 30°.

Example 1 Workpiece with tilted operating plane.

N10 G17
 N20 G54
 N30 M55
 N40 G7 L1=1
 N..
 N100 G81 Y1 Z-30
 N110 G79 X40 Z0
 N120 G79 X90
 N..
 N200 G0 X130 Z50
 N210 G93 X130
 N220 G7 B5=30 L1=2 L50
 or
 G7 B5=30 L1=1

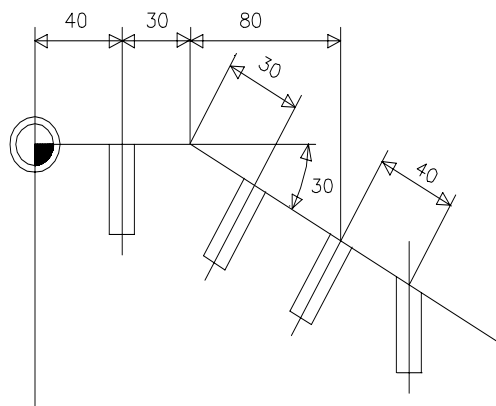
N230 G79 X30 Z0
 N240 G79 X70
 N..
 N300 G0 Z50
 N310 G7 L1=2 L50 or L1=1

Define operating plane
 Zero point shift
 Deselect ion M53/M54
 Reset G7

Drill cycle definition
 Drill the first hole in the horizontal plane
 Drill the second hole in the horizontal plane
 Other movements in the horizontal plane
 Tool is set at the safety distance
 Zero point is set at the start of the tilted operating plane
 Define new operating plane
 B5=30 Angle of rotation
 L1=1 Interpolate only the rotary axes, not the linear axes.
 L1=2 Tool/table is rotated about the tooltip
 L50 Extra length oversize in the direction of the tool. In this way, the tool is rotated about the zero point. The distance from the tooltip to the zero point is 50 mm

Drill the first hole in the tilted operating plane
 Drill the second hole in the tilted operating plane
 Other movements in the tilted operating plane
 Tool is set at the safety distance
 Reverse rotation in the horizontal plane and close G7

Example 2 Workpiece using tilted operating plane.



N10 G17	Defining operating plane
N20 G54	Zero point shift
N30 M55	Deselect ion of M53/M54
N40 G7 L1=1	Reset G7
N..	
N100 T1 M6	Tool change (drill)
N110 G81 Y1 Z-30	Define the drilling cycle
N120 G79 X40 Z0	Drilling of a hole in the horizontal plane
N..	Other movements in the horizontal plane
N200 T2 M6	Tool change (mill)
N210 X70 Z50	Tool is set at the safety distance
N220 G93 X70	Define the zero point of the new operating plane
N230 G7 B5=30 L1=2 L50	Define new operating plane
	B5=30 Angle of rotation
	L1=2 Tool/table is rotated about the tooltip
	L50 Extra length oversize in direction of the tool. In this way the tool is rotated about the zero point. The distance of the tooltip from the zero point is 50 mm.
N240 G1 X0 Z0	Positioning of the mill perpendicular to the tilted plane
N250 X150	Mill in the tilted plane
N..	Other movements in the tilted operating plane
N300 T1 M6	Tool change (drill)
N310 G79 X30 Z0	Drilling the first hole in the tilted operating plane
N320 G92 X=80:cos(30)	Incremental zero point shift
N330 G79 X0 Z0	Drilling the second hole in the tilted operating plane
N..	Other movements in the tilted operating plane
N400 G92 X=40	Incremental zero point shift
N410 G0 X0 Z50	Tool is set at the safety distance
N420 G7 B5=0 L1=2 L50	Deselect "Tilt operating plane". Reverse rotation in the horizontal plane
	B5=30 angle of rotation
	L1=2 Tool/table is rotated about the tooltip
	L50 Extra length oversize in the direction of the tool. In this way, the tool is rotated about the zero point. The distance of the tooltip from the zero point is 50 mm
N430 G79 X0 Z0	Drilling the third hole in the horizontal operating plane
N..	Other movements in the horizontal operating plane
N500 M30	Program end.

Example 3 Determination zero point with G7 and G54 I[Nr.]

Procedure:

G54 I[Nr.] will be active, but B4= must be zero.

Tilting of von G7 with MDI (for example: B5=45 C5=-45 L1=1 (Only rotation of the rotary axis))

Place the measure probe in the centre of the hole.

Start program N54

N54 (Program for determination zero point in G7 plane)

N1 E1=35 E1=zero point number.

N2 E2=20 E2=hole radius.

N2 G54 I=E1 X0 Y0 Z0 A0 B0 C0 B4=0 Setting zero point shifts on zero.

N3 G51

N4 G53 Cancel all zero point shifts.

N5 G326 X7=50 Y7=51 Z7=52 Query und storing actual position of the measure probe.
E50= X, E51=Y, E52=Z.

N6 M27 M-function for activating the measure probe.

N7 (Measuring in G7 plane, first measuring in positive X-direction)

N9 G0 X=E50+E2-5 Y=E51 Z=E52 To start position. 5 mm distance to hole border. Collision
when E2=<5.

N7 G145 X=E50+E2+10 Y=E51 Z=E52 L0 X7 F2 E40 I3

X, Y, Z End position, X is border+10.

L0 Measure by contact.

X7=49 Measured position in E49.

F2=50 Measure feed.

E40 Measure status in E40.

I3=0 Status-control on.

Post measuring distance is 10 mm.

N8 G29 E41 E40<>1 N= 24 If no point is measured, jump to program-end.

N9 G0 X=E50-E2-5 Y=E51 Z=E52

To start position second measuring in negative X-direction.

N10 G145 X=E50-E2-10 Y=E51 Z=E52 L0 X7=48 F2=50 E40 I3=0

N11 G29 E41 E40<>1 N= 24 If no point is measured, jump to program-end.

N13 G0 X=E50 Y=E51+E2-5 Z=E52

To start position third measuring in positive Y-direction.

N14 G145 X=E50 Y=E51+E2-5 Z=E52 L0 Y7=47 F2=50 E40 I3=0

N15 G29 E41 E40<>1 N= 24 If no point is measured, jump to program-end.

N16 G0 X=E50 Y=E51-E2+5 Z=E52

To start position fourth measuring in negative Y-direction.

N17 G145 X=E50 Y=E51-E2-10 Z=E52 L0 Y7=46 F2=50 E40 I3=0

N18 G29 E41 E40<>1 N= 24 If no point is measured, jump to program-end.

N19 (Measuring perpendicular to auf G7 plane, fifth measuring in negative Z-direction.

N19 G0 X=E49+5 Y=E51 Z=E52+5 To start position at the top of the material.

N20 G145 X=E49+5 Y=E51 Z=E52-10 L0 Z7=45 F2=50 E40 I3=0

N21 G29 E41 E40<>1 N= 24 If no point is measured, jump to program-end.

N22 G54 I=E1 X=(E49+E48):2 Y=(E47+E46):2 Z=E45

Setting zero point. X, Y, und Z must be given. The
coordinates will be converted and after that storing in the
original machine coordinates system)

N23 G0 X0 Y0 Z0

Go to hole centre point. The display coordinates are all
zero.

N24 M28

M-function for deactivating measure probe.

N25 M30

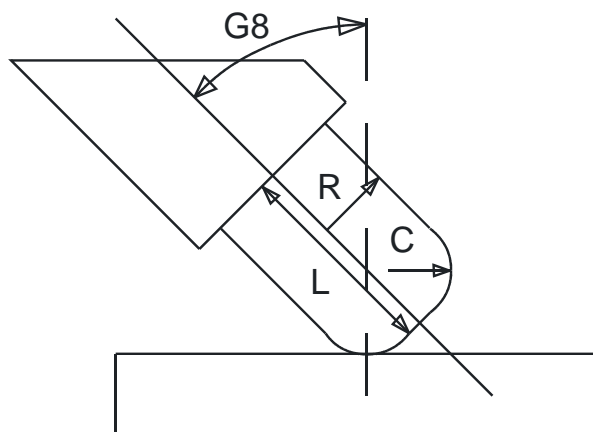
5.7 G8 Tilting tool orientation

To program a swivelled tool for four or five-axis machines.

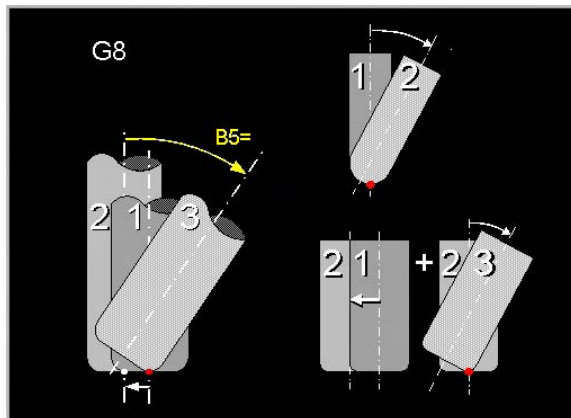
With the function "Swivel tool" the tool axis can be set obliquely relative to the machining plane. This makes angle milling possible and substantially improves cutting conditions and thus surface quality. The programming of G8 is identical to G7. G7 should therefore also be read.

Format

G8 {A5=.. or A6=..} {B5=.. or B6=..} {C5=.. or C6=..} {A7=..} {B7=..} {C7=..} {L} {L1=..} {L2=} {L3=..} {F6=..} {F}



L, R and C from the tool table.



G Tilting tool orientation
 L Tool length offset
 A5= Angle of rotation absolute
 B5= Angle of rotation absolute
 C5= Angle of rotation absolute
 A6= Angle of rotation incremental
 B6= Angle of rotation incremental
 C6= Angle of rotation incremental
 B7= E-par. for position in B
 C7= E-par. for position in C
 L1= 0=No move., 1=rot.axes, 2=tooltip
 L2= -/+1,2,3 = Neg/Pos A,B,C angle
 L3= Radius compensation (0=on, 1=off)
 F6= Block feed

Notes and usage

Modality

This function is modal and stay active until another G8 is programmed.

G functions not permitted within G8

The following G functions are not permitted when G8 is active:

G6, G19, G40, G41, G42, G43, G44, G66, G67, G141, G180, G182, G339

solid angle

A5=, B5=, C5= Defines the absolute angle by which the tool direction rotates relative to the 'normal' tool axis.

A6=, B6=, C6= Defines the incremental angle by which the tool direction rotates relative to the 'normal' tool axis.

Value between -359.999 and 359.999 [degrees]

Redefine tool direction

The rotation of the tool direction can be defined in two ways:

- Programming with A5=, B5= or C5= parameters. This defines the absolute rotations about the corresponding positive axes. The rotations are calculated as follows:
 1. the active G8 rotation is cancelled
 2. C5= rotation about the positive Z axis fixed with respect to the machine
 3. B5= rotation about the positive Y axis
 4. A5= rotation about the positive X axis
- Programming with A6=, B6= or C6= parameters. This defines the incremental rotations about the corresponding current positive axes. The rotations are calculated as follows:
 1. C6= rotation about the current G8 positive Z axis
 2. B6= rotation about the current G8 positive Y axis
 3. A6= rotation about the current G8 positive X axis

Programming is independent of the machine configuration. The plane rotation is calculated relative to the current zero point. The motion is dependent on the machine configuration.

Scanning a calculated angular position

A7=, B7=, C7= Contains the number of the E parameter in which the calculated angle of the corresponding rotary axis is set.

Alternative tilting possibilities within moving range of the machine

The CNC checks, which tilting possibilities within the moving range of the rotary axes are possible (to the left or to the right).

- No tilting possibilities, than error message is given (P307)
- By only one tilting possibility this will be executed.
- By two tilting possibilities, those with the shortest movement will be executed (L2=0 or not programmed). The shortest movement is not always possible.

With the address L2= can be controlled, which tilting possibilities must be executed. By L2=1/2/3 the A/B/C-axis is positioning so, that a positive angle will be reached. By negative L2= a negative angle will be reached.

Feed

F6= is a local feed which is only active in the record in which it is programmed. In this case, it is the rotation of the tool. F is the normal feed and also applies to the subsequent records.

Swivel motion

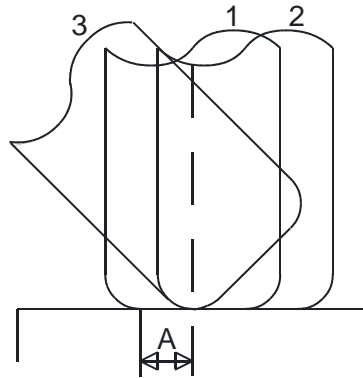
The G8 swivel motion is performed by interpolating with feed (F6=). It swivels the tool axis onto the defined plane. Which axes move depends on the type of motion L1= :

- L1=0 The rotary axes do not move (default).

Note: The swivel motion can be programmed or performed manually by means of the E parameters that are loaded with A7=, B7= or C7=.

- L1=1 Only the rotary axes swivel, while the linear axes do not move.
- L1=2 The rotary axes swivel and the linear axes perform a movement. This means that the contact point position remains X, Y, Z.

If the contact point lies on the tool corner radius, the movement is only a rotation. If the contact point is the tooltip and the corner radius (C) is smaller than the tool radius (R), a compensating movement occurs so that the contact point is shifted from the tooltip to the corner radius.



With cylindrical cutters (with corner radius $C < \text{cutter radius } R$), the following particular point applies: When swivelling from the vertical (1) to the oblique (2--> 3) position or vice versa, the contact point moves from the centre of the cutter to the corner radius (A) and vice versa. A compensating movement at the tooltip ensures that the current contact position X, Y, Z remains unchanged.

Caution: The movements when applying/cancelling the tool correction within G8 may result in a collision. It is the responsibility of the programmer (operator) to avoid this.

Toollength allowance

If the swivel motion takes place about the tool contact point ($L1=2$), L defines an extra allowance in the tool direction between the rotation point and the tooltip.

Toolradius-correction ($L3=$)

Dependent of the toolradius-correction ($L3=$) the radiuscorrection will be calculated.

When $L3=1$ the radius (R) and the cornerradius (C) will be not considered. The tool turns around the tooltip and a compensation movement will not be made.

When $L3=0$ the radius (R) and cornerradius (C) will be considered. The tool turns and a compensation movement will be made.

Default $L3=0$.

Tool correction

During the function "swivel tool" (G8) the values L, R and C, dependent of the toolradius-correction ($L3=$), for the tool are corrected.

This G8 tool correction is independent of G40, G41, G42, G43, G44 and is always effective.

At the beginning and end of the tool correction, a compensation movement is carried out if the corner radius (C) is smaller than the tool radius (R).

If the tool dimensions (L, R, C) change with G8 active, the current position of the linear axes is recalculated.

Turning off the G8 function

G8 remains active until it is cancelled. G8 is cancelled by programming G8 without angle parameters.

G8 is not cancelled by M30 or <program abort>. After the control is turned on, G8 is still active. After search for reference points or <CNC reset> G8 is cancelled.

Note: At the start any program that uses G8, we recommend that a G8 be programmed with no parameters. This ensures that the tool direction is always reset as the program is starting up (abort with swivelled tool and new start). Without this G8 at the beginning, the first part of the program is executed in the swivelled instead of the unswivelled plane.

The programming is similar to programming with G7/G17/G18 - different zero points or different tools.

Configuration

Swivel tool (G8) can be used for machines where a kinematic model is defined and entered. See description of the kinematic model.

Graphics

The G8 has no effect on the graphics.

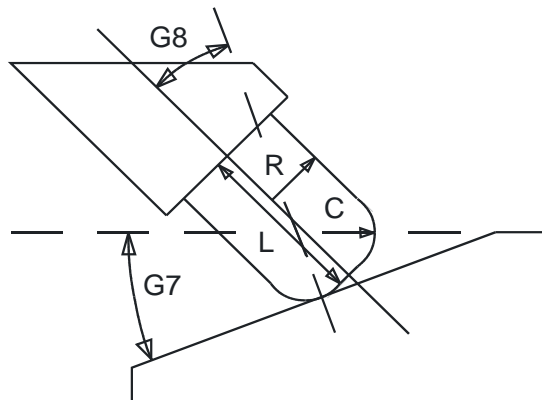
Display

When G8 is active, a yellow icon is displayed in the display behind the tool number.

A small 'p' on the right, next to the 'axis letters', is used to display whether the position of the tooltip is displayed or the position in machine coordinates.

Example

Workpiece with oblique machining plane and oblique tool.



N10 G17

Define machining plane

N20 G54

Zero point offset

N30 M55

Deselect M53/M54

N40 G7 L1=1

Reset G7

N50 G8 L1=1

Reset G8

..

N100 G0 X130 Z50

Tool set to safety distance

N110 G93 X130

Zero point set to the beginning of the swivelled machining plane.

N120 G7 B5=-30 L1=2

Define new oblique position of the tool.

B5=-30 Angle of rotation

L1=2 Tool/table revolves about the tooltip

N130 G8 B5=30 L1=2

Define new oblique position of the tool.

B5=30 Angle of rotation

L1=2 Tool rotates about the tooltip and a compensation movement is performed.

..

N200 G8

Turn tool perpendicular to the machining plane again (rotary and compensation movement).

N210 G7 L1=2

Rotate back to the horizontal plane.

5.8 G9 Define pole position (size reference point)

To program a pole point. When a pole point has been programmed, program blocks with pole programming (angle and length) no longer relate to the zero point, but to the pole point most recently programmed.

The pole point is programmed as a function of the modally valid system of measurement G90/G91. Furthermore, the pole point may be programmed wordwise absolute, incrementally or combined absolute/incrementally.

Format

G17 active: G9 X.. Y.. {X90=...} {X91=...} {Y90=...} {Y91=...}

G18 active: G9 X.. Z.. {X90=...} {X91=...} {Z90=...} {Z91=...}

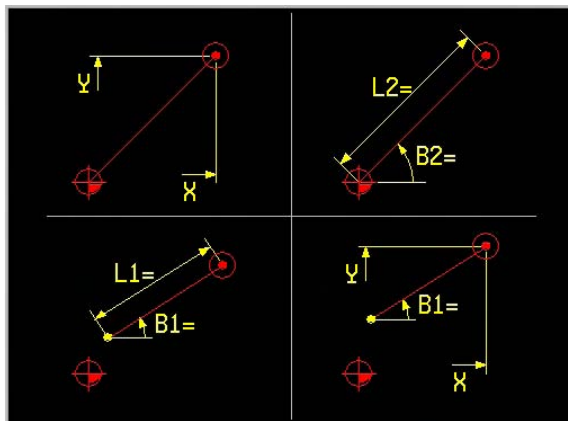
G19 active: G9 Y.. Z.. {Y90=...} {Y91=...} {Z90=...} {Z91=...}

Deactivate pole (identical with workpiece zero point) G9 X0 Y0

Pole point in polar coordinates (G17, G18, G19 active):

absolute: G9 B2=.. L2=..

incremental: G9 B1=.. L1=..



```
G Define pole position
X Pole coordinate
Y Pole coordinate
Z Pole coordinate
B1= Angle
B2= Polar angle
?90= Pole coordinate abs. (X,Y,Z..)
?91= Pole coordinate incr. (X,Y,Z..)
L1= Path length
L2= Polar length
```

Notes and usage

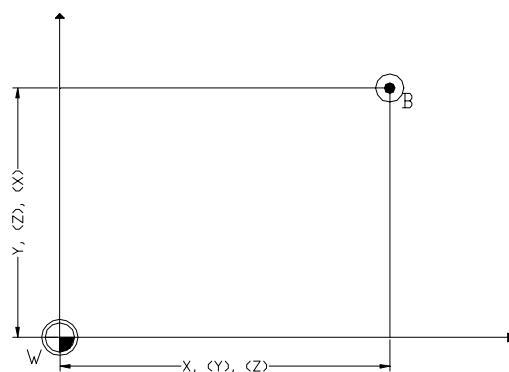
Modality

This function is modal with G0, G1, G2, G3 and G6.

Pole point in Cartesian coordinates:

Pole point in absolute coordinates:

The programmed coordinates relate to the workpiece zero point.

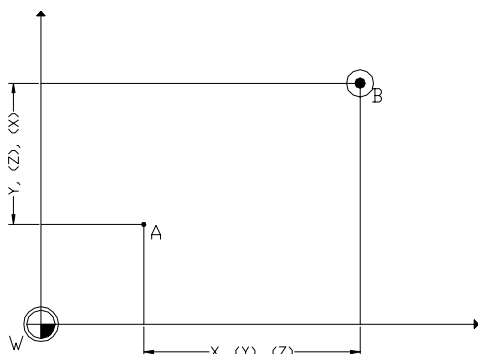


B = Polpunkt

N.. G9 X.. Y..

Pole point in incremental coordinates:

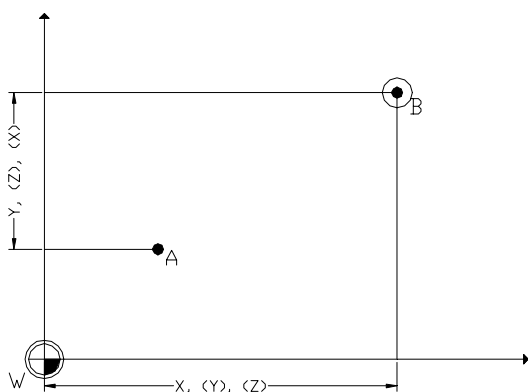
The programmed coordinates relate to the actual position



A = existing pole point

B = new pole point

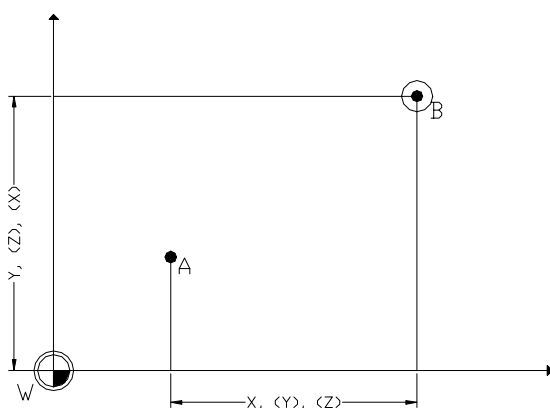
N... G9 X91=... Y91=...

Pole point in combined absolute/incremental coordinates:

A = existing pole point

B = new pole point

N... G9 X... Y91=...



A = existing pole point

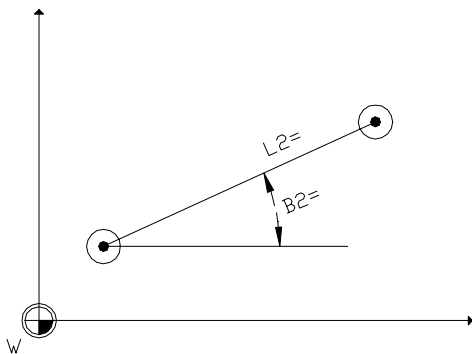
B = new pole point

N.. G9 X91=.. Y..

Pole point in polar coordinates (G17, G18, G19 active):

Pole point in absolute polar coordinates:

The polar coordinates B2= and L2= relate to the most recently active pole point.

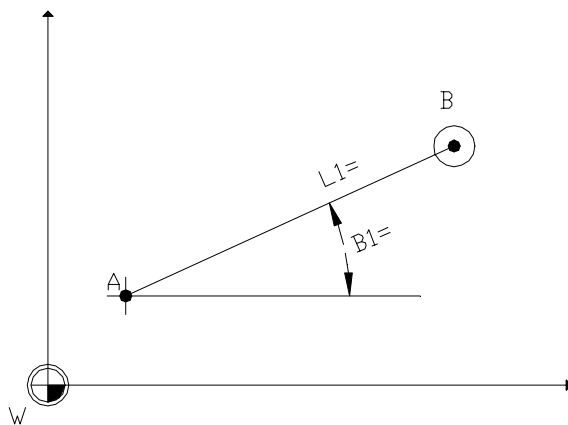


A = existing pole point
B = new pole point

N.. G9 B2=.. L2=..

Pole point in incremental polar coordinates:

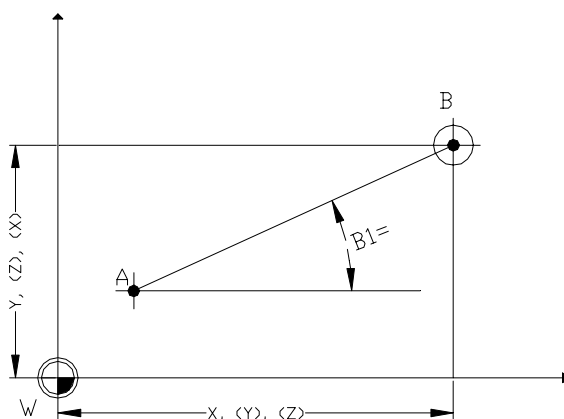
The polar coordinates B1= and L1= relate to the actual position.



A = end point of last movement
B = new pole point

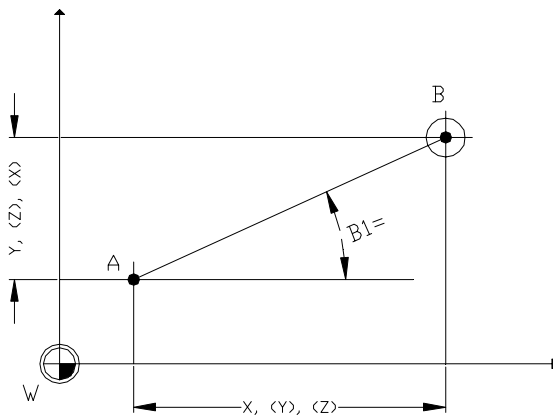
N.. G9 B1=.. L1=..

Combined programming: Cartesian absolute/polar:



A = existing pole point
B = new pole point

N.. G9 X.. B1=..

Combined programming: Cartesian absolute/polar:

A = existing pole point
B = new pole point

N.. G9 X91=.. B1=..

- polar definitions are allowed in the active working plane only
- before the G9 block is called, the pole point is at the workpiece zero point (pole point = 0)
- the pole point is modally active
- the pole point may be redefined indefinitely
- the pole point is zeroed (0) when changing the plane using G17, G18, and G19

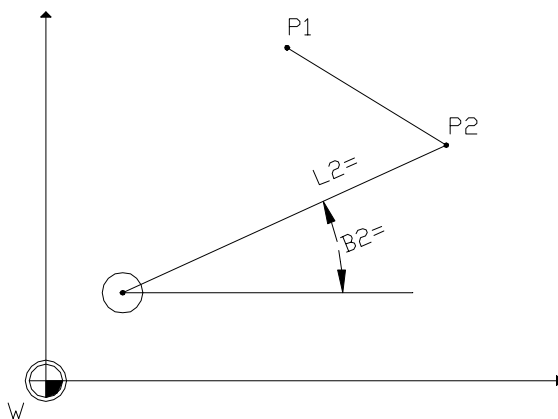
Polar definition of end point:

During absolute polar programming the polar lengths L2= and L3= and polar angles B2= and B3= no longer relate to the zero point, but to the pole point.

If no pole point has been defined, the pole point = 0 (zero) and therefore equals the active zero point.

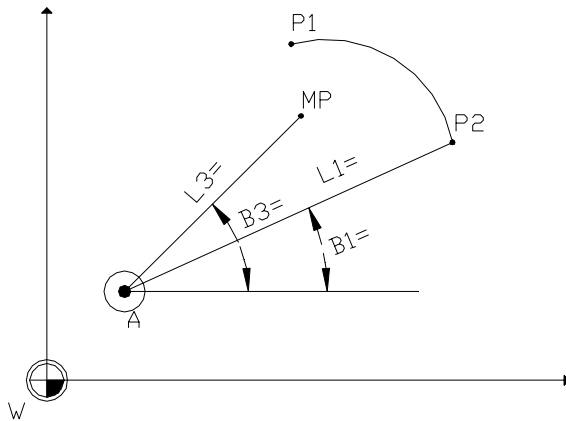
Pole point definition

Pole points with pole can be defined in the following G-functions:
G0, G1, G40, G44, G61, G62, G77, G78, G79, G145



Polar circle definition

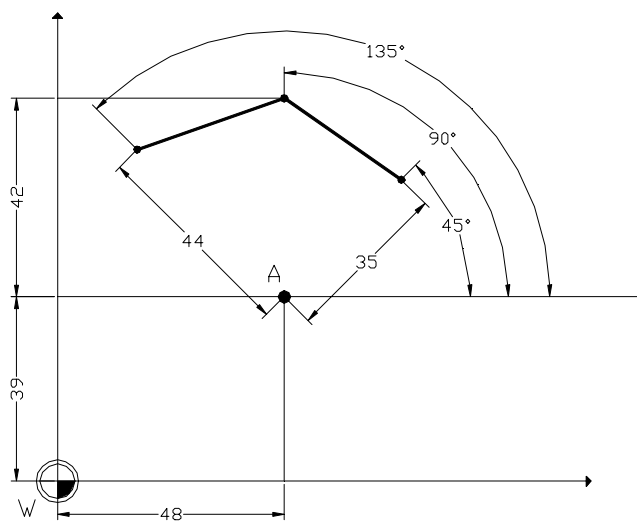
Polar programming with pole point of the centre point and end point is possible in G2 and G3 blocks.



ICP/Geometry calculation G64

G1, G2 and G3 blocks with B2=, B3= and L3= programming can be programmed in G64 and ICP. These blocks relate to the active pole point. The pole point can only be changed in G64 and not in ICP.

Example



A = new pole point

```
N30 G9 X48 Y39
N40 G1 B2=135 L2=44
N50 G1 B2=90 L2=42
N60 G1 B2=45 L2=35.
```

Definition of new pole point

Definition of end point coordinates related to new pole point

5.9 G11 Linear chamfer or rounding cycle

Note Use of this function is limited only to programs made on earlier control systems.

The operator can easily make programs requiring geometry calculations, using Interactive Contour Programming (ICP).

Format

One point geometry (XY-plane)

G11 X... Y... {K...} {R...} {F...}

G11 B... L... {K...} {R...} {F...}

Two point geometry (XY-plane)

G11 X... Y... X1=... Y1=... {K...} {R...} {K1=...} {R1=...} {F...}

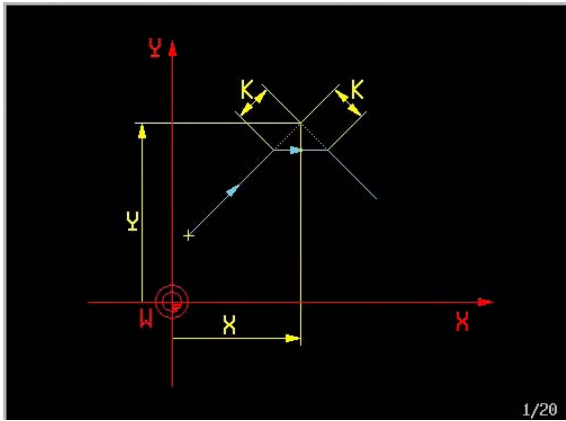
G11 B... L... X1=... Y1=... {K...} {R...} {K1=...} {R1=...} {F...}

G11 X... Y... B1=... L1=... {K...} {R...} {K1=...} {R1=...} {F...}

G11 B... L... B1=... L1=... {K...} {R...} {K1=...} {R1=...} {F...}

Two line geometry (XY-plane)

G11 B... X... Y.. B1=... {K...} {R...} {K1=...} {R1=...} {F...}

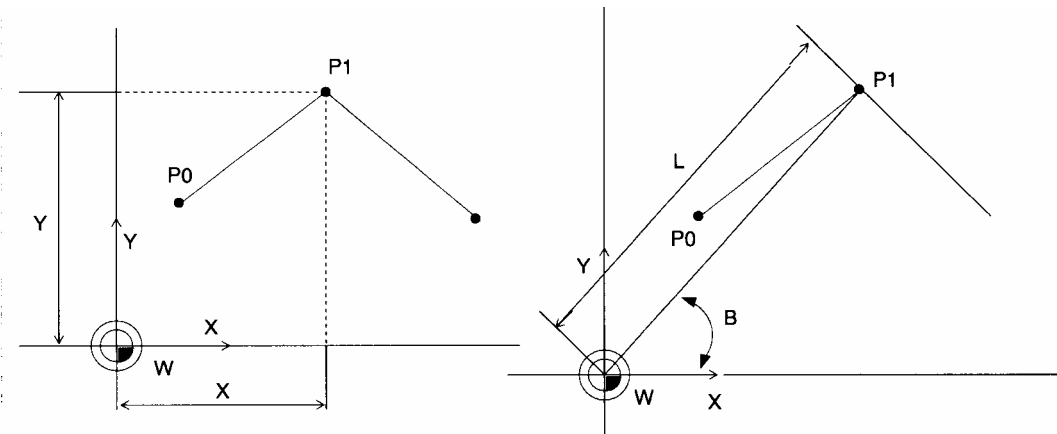


G	Linear chamfer rounding cycle
X	Endpoint coordinate
Y	Endpoint coordinate
Z	Endpoint coordinate
B	First angle
K	First chamfer length
L	First length
R	First rounding radius
X1=	Arbitrary endpoint coordinate
Y1=	Arbitrary endpoint coordinate
B1=	Second angle
K1=	Second chamfer length
L1=	Second length
P1=	Point definition number
R1=	Second rounding radius

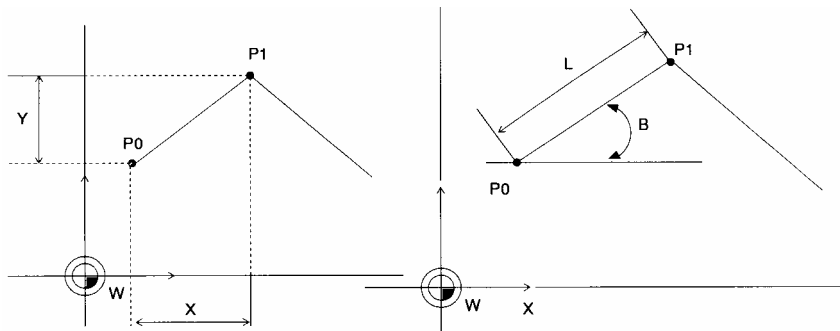
1. One Point Geometry

To program in one block:

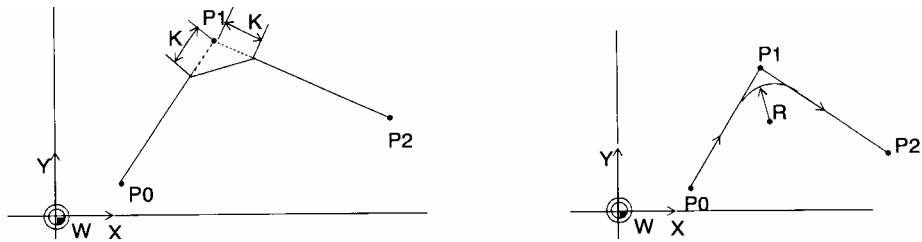
- The end point of a linear movement
- If required, a symmetrical chamfer or rounding between this movement and the next linear movement.



Absolute coordinates (G90)



Incremental coordinates (G91)



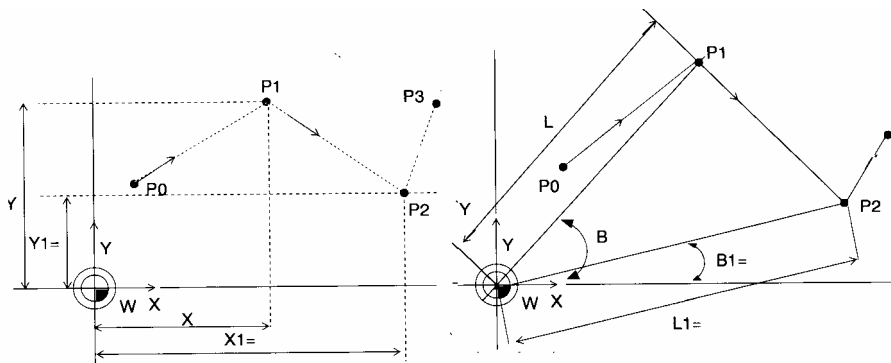
Chamfer rounding with one point geometry

2. Two Point Geometry

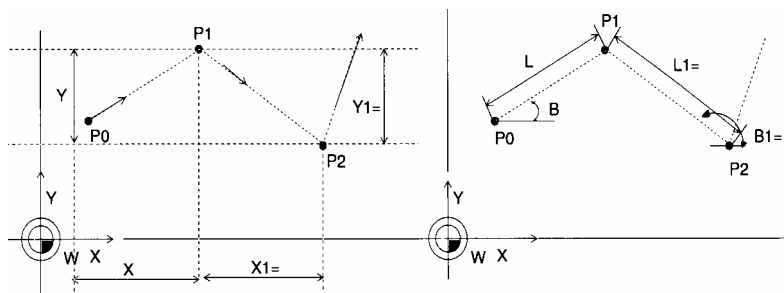
To program in one block:

- The end points of two separate linear movements
- If required, a symmetrical chamfer or rounding between these movements
- If required, a symmetrical chamfer or rounding between the last movement and the next linear movement.

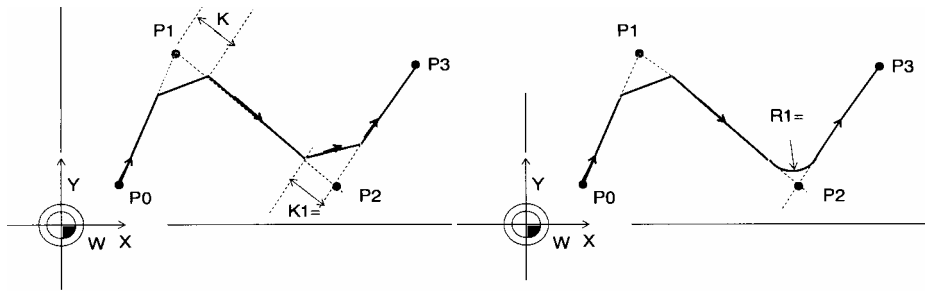
Chamfers or roundings with two-point geometry.



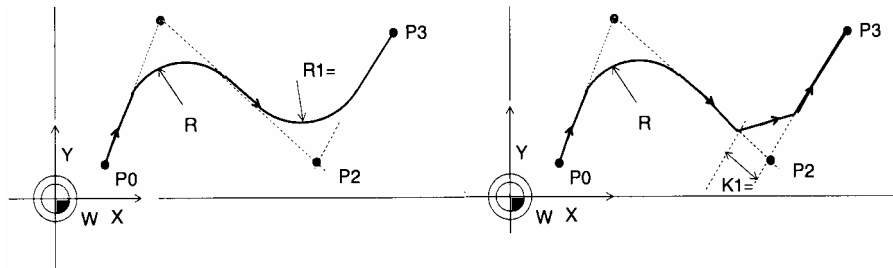
Absolute coordinates (G90)



Incremental coordinates (G91)



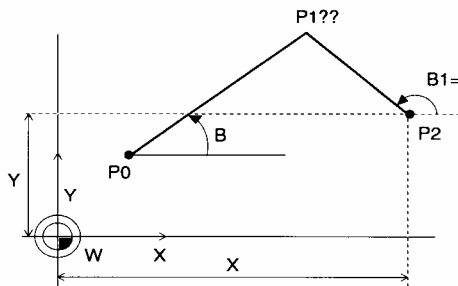
Chamfers or roundings with two-point geometry



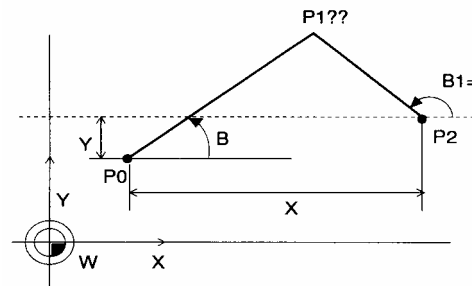
3. Two Line Geometry

To program in one block two separate linear movements:

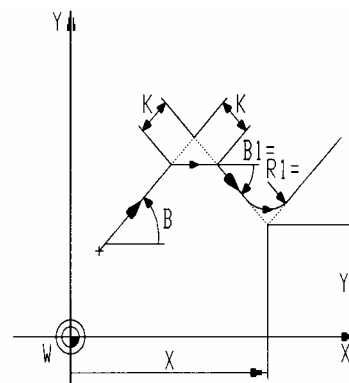
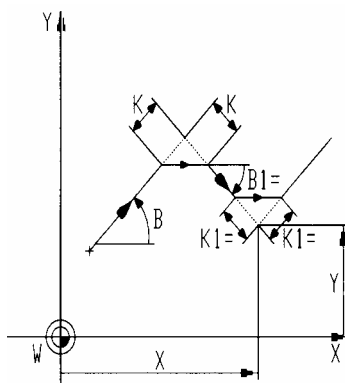
- The first linear movement with the angle with the main axis
- The second linear movement with the end point and the angle with the main axis
- If required, a symmetrical chamfer or rounding between these movements
- If required, a symmetrical chamfer or rounding between the last movement and the next linear movement.



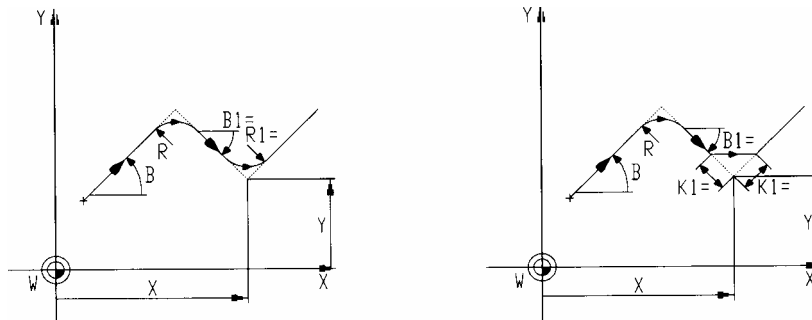
Absolute coordinates (G90)



Incremental coordinates (G91)



Chamfers or roundings with two-line geometry

**One point geometry**

X, Y, Z	Linear axis coordinates (absolute/incremental)
P	Point definition number.
P1=	Point definition number.
B	G90 active: Angle the line through the datum point W and the end point makes with the X-axis (G17 and G18) or -Z-axis (G19)
	G91 active: Angle the line makes with the X-axis (G17 and G18) or -Z-axis (G19)
L	G90 active: length measured from the datum point W to the end point
	G91 active: length measured from the last tool position to the end point

Two point geometry

X, Y, Z	Linear axis coordinates of the first point (absolute/incremental). No tool axis allowed.
P1=	Point definition number of the first point.
B	G90 active: Angle the line through the datum point W and the first end point makes with the X-axis (G17 and G18) or -Z-axis (G19)
	G91 active: Angle the first line makes with the X-axis (G17 and G18) or -Z-axis (G19)
L	G90 active: length measured from the datum point W to the first endpoint
	G91 active: length measured from the last tool position to the first end point
X1=, Y1=, Z1=	Linear axis coordinates of the second point (absolute/incremental). No tool axis allowed.
P2=	Second point definition number.
B1=	G90 active: Angle the line through the datum point W and the second end point makes with the X-axis (G17 and G18) or -Z-axis (G19)
	G91 active: Angle the second line makes with the X-axis (G17 and G18) or -Z-axis (G19)
L1=	G90 active: length measured from the datum point W to the second endpoint
	G91 active: length measured from the first end point to the second endpoint

Two line geometry

X, Y, Z	Linear axis end point coordinates of the second line (absolute/incremental). No tool axis allowed.
P	Point definition number of the end point of second line.
P1=	Point definition number of the end point of second line.
B	Angle the first line makes with the X-axis (G17 and G18) or -Z-axis (G19)
B1=	Angle the second line makes with the X-axis (G17 and G18) or -Z-axis (G19)

Words for chamfer or rounding in the three cases

K	First chamfer length
R	First rounding radius
K1=	Second chamfer length
R1=	Second rounding radius

Notes and usage**Feedrate**

All movements in a G11-block use the last programmed feedrate if a New feedrate is not stated in the G11-block.

Next movement after a G11 block

If a second chamfer (K1=) or a second rounding (R1=) has been programmed, the block following the G11-block must contain either a G1 or G11-function.

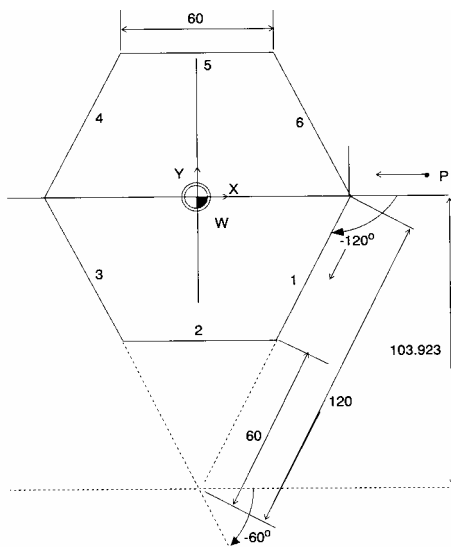
If a G1-block has been programmed following a G11-block, both end point coordinates (e.g. X. and Y.) must be stated.

Tool axis programming with a G11

With G11 tool axis programming is not allowed

Restriction

1. The G11 function is not allowed once the geometry is activated (G64 active).
2. The G11 function is not allowed for defining a pocket or island contour. (G200..G208)
3. G11 is not allowed with a programmed tool axis. In case programs are started with above mentioned programming combination, the programmer may encounter operational errors P01 and/or P34 at execution of the program.

Examples**Example 1** One point geometry

The regular hexagon has to be milled on the outside of the workpiece surface. The one point geometry with angle is used. The sides 2 and 4 are programmed as chamfers.

N9010

N1 G17 T1 M6

N2 G0 X100 Y10 Z-10 S1000 M3

N3 G1 F300

N4 G43 X60

N5 G41 Y0

N6 G11 B-90 L103.923 K60

N7 G11 B150 L103.923 K60

Activate the main plane. Load the tool

Start the spindle, move tool to point P and to depth.

Set feedrate to 300 mm/min.

Move the tool to the corner of the hexagon.

Set radius compensation LEFT.

Mill along sides 1 and 2.

Programmed is:

- the intersection point of sides 1 and 3
- the chamfer (K-word) around this point.

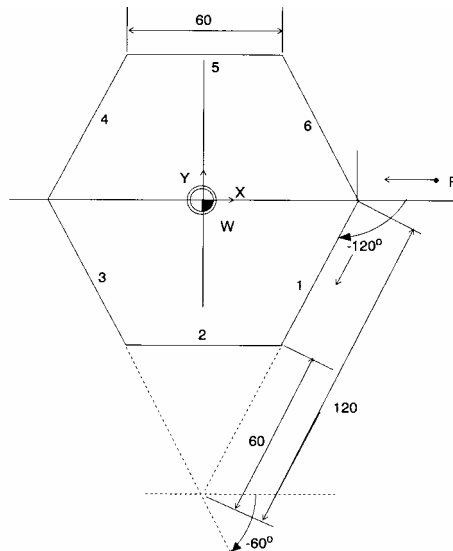
Mill along sides 3 and 4.

Programmed is:

N8 G11 B60 L60
N9 G11 B0 L60
N10 G40
N11 G1 X100 Y10
N12 G0 Z100 M30

- The intersection point of sides 3 and 5
 - The chamfer (K-word) around this point.
- Mill along side 5.
Mill along side 6.
Cancel the radius compensation.
Move tool away from part.
Retract the tool and end of program.

Example 2. Two point geometry



The regular hexagon has to be milled on the outside of the workpiece surface. The two-point geometry with angles and increments is used. The sides 2 and 5 are programmed as chamfers.

N9011
N1 G17 T1 M6
N2 G0 X100 Y10 Z-10 S1000 M3
N3 G1 F300
N4 G43 X60
N5 G41 Y0
N6 G91

- Activate the main plane. Load tool 1
- Start the spindle, move tool to point P and then to depth.
- Enter the linear movement and set the feedrate.
- Move the tool to the corner of the hexagon.
- Set radius compensation LEFT.
- Activate the incremental mode. The length values in the next blocks are measured from the previous tool position.

N7 G11 B-120 L120 K60 B1=-120 L1=120 Mill along sides 1, 2 and 3.

Programmed is:

- The intersection point of sides 1 and 3 (B and L),
- The chamfer (K-word) around this point
- The end point of side 3 (B1= and L1=).

N8 G11 B60 L120 K60 B1=-60 L1=120 Mill along sides 4, 5 and 6.

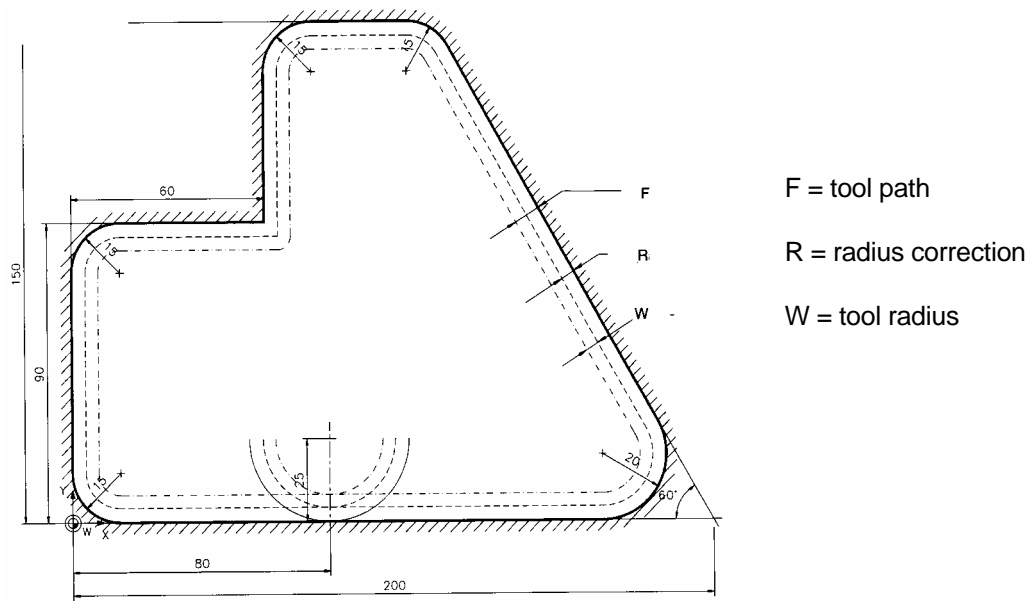
Programmed is:

- The intersection point of sides 4 and 6 (B and L),
- The chamfer (K-word) around this point
- The end point of side 6 (B1= and L1=)

N9 G40
N10 G90
N11 G1 X100 Y10
N12 Z10 M30

- Cancel radius compensation.
- Set the absolute mode.
- Move tool away from workpiece.
- End of program.

Example 3. Two line geometry



The inside pocket can be programmed using G11-function with two line geometry elements.

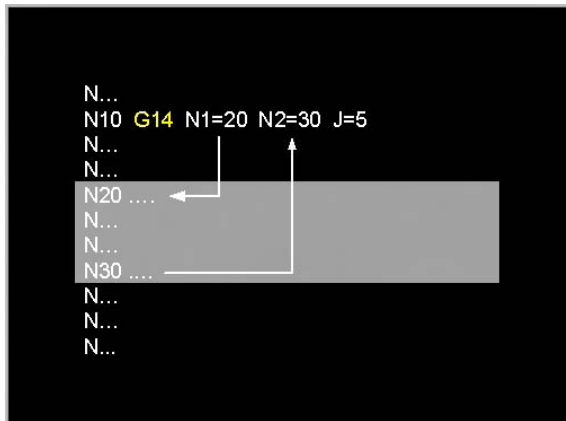
N9012	
N1 G17	Activate XY-plane (G17)
N2 X80 Y25 Z0 T1 M6	Load tool T1 (Mill diameter 10 mm). Move the tool to point B and above the workpiece.
N3 G1 Z-10 F500 S1000 M3	Start the spindle and feed to depth.
N4 G43 X105	Move the tool to the starting point of the entering-circle.
N5 G42	Set radius compensation RIGHT.
N6 G2 X80 Y0 R25 F300	Move to the contour via the entering-circle.
N7 G11 X0 Y90 B180 B1=90 R15 R1=15	Mill - along the X-axis, (B0)
	- Along the radius, (R15)
	- Along the Y-axis, (B1 =90)
	- Along the second radius (R1 =15).
N8 G11 X60 Y150 B0 B1=90 R1=15	Mill - parallel to the X-axis, (B0)
	- Parallel to the Y-axis, (B1 =90)
	- Along the second radius (R1 =15)
N9 G11 X200 Y0 B0 B1=120 R15 R1=20	Mill - parallel to the X-axis, (B0)
	- Follow the first radius, (R15)
	- Mill along the slope of 60 degrees (B1=120)
	- Follow the second radius (R20).
N10 G1 X80 Y0	Mill along the X-axis to the starting point of the circle for leaving the contour.
N11 G2 X55 Y25 R25	Exit the contour with a circular movement.
N12 G40	Cancel the radius compensation.
N13 G0 Z200 M30	Retract the tool. End of program

5.10 G14 Repeat function

To repeat the execution of a specified number of blocks within a partprogram or subprogram.

Format

G14 N1=... {N2=...} {J...} {K...}



G Repeat function
J Number of repeats
K Repeat decrement
N1= Repeater begin block
N2= Repeater end block

Notes and usage

Block numbers of repeat sequence (N1=, N2=)

These block numbers must be in the same partprogram or subprogram.

If N2= is not programmed, only the block indicated by N1 = is repeated the specified number of times.

Order of blocks to be repeated

The order of executing the blocks in the repeat sequence must be the same as the order originally programmed. So in the program block N1=.. must be before block N2=..

Number of repeats (J)

Instead of J an E-word can be used.

The number of repeats is programmed with the J-word. The J-word is not necessarily an integer value. The integer part, thus the part before the decimal point, is used as the number of repeats.

When no number of repeats is programmed (no J-word is present), the sequence is repeated only once.

Repeat decrement (K)

The K-word allows the value of the J-word to be recalculated and used as the condition for repeating.

If the K-word is not programmed, the value of the J-word is reduced by 1 after every repeat.

If $K > 0$, the value is used to reduce the value of the J-word. If e.g. K5 were programmed, 5 would be subtracted from the value of the J-word after every repeat. As long as the J-word is greater than 0, a repeat is executed.

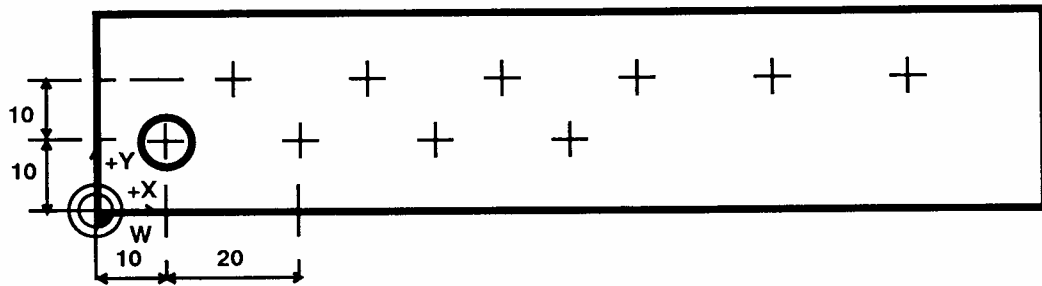
If $K \leq 0$ is programmed, an error message is displayed.

Nesting of repeats

A repeating block sequence may be included in another repeating block sequence; this can be done four times.

Continuation after the repeat

Once the repeats are executed, the program continues with the block after the G14.

Example Programming a repeat function

N1234

N1 G195 X-10 Y-10 Z10 I160 J50 K-30

N2 G99 X0 Y0 Z0 I140 J30 K-10

N3 G17

N4 T1 M6

N5 G81 Y5 Z-11.5 F100 S2000 M3

N6 G79 X10 Y10 Z0

N7 G79 L1=20 B1=0

N8 G14 N1=7 J2

N9 G92 X10 Y10

N10 G14 N1=6 N2=8

N11 G14 N1=7 J2

N12 G93 X0 Y0

N13 G0 Z200

N14 M30

Set up graphic window

Set up graphic (material)

Define the main plane.

Load tool 1 with a drill diameter of 10 mm.

Define fixed drilling cycle and start the spindle

Drill holes

Drill holes with Polar coordinates.

Program block N7 will be repeated twice.

Absolute zero point shift

Program block N6..N8 will be repeated once.

Program block N7 will be repeated twice.

Absolute zero point shift.

Retract the tool to Z200.

End of program.

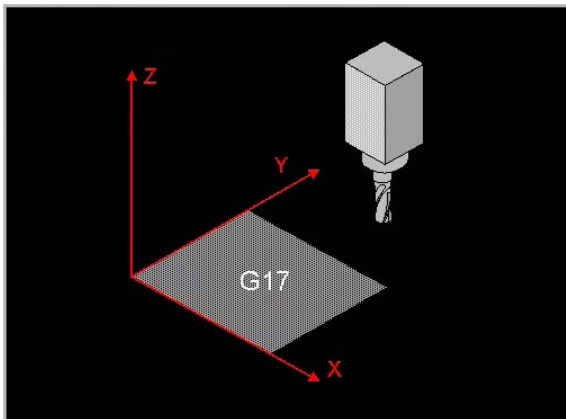
5.11 G17 Mainplane XY, tool Z

The main spindle of the machine tool determines the position of the tool axis. With G17 is defined that the tool axis is the Z-axis and the main plane for milling operations the XY-plane.

For turning mode refer to chapter "Mainplane for turning mode".

Format

G17



G Mainplane XY, tool Z

Notes and usage

Modality

This function is modal with G18 und G19.

Default plane

When switching on the machine or after a CLEAR CONTROL the machine activates automatically due to MC11 (0=G17, 1=G18, 2=G19) a PLANE.

The last selected PLANE is active when the machine is switch on normally.

Operations in the plane

Calculations for radius compensation, the geometry (G64), polar coordinate, milling cycles; the pocket cycle, etc. are performed in the current plane. Thus when G17 is active the XY-plane.

Operations in the tool axis

Toollength compensation and the fixed cycles for hole operations use the current tool axis. Thus when G17 is active the Z-axis.

Angular head

When an angular head is fitted, the axis configuration of the machine tool remains unchanged. So the tool can be in either the Y- or X-axis.

With the function G18 or G19 is programmed in which axis the tool is standing and which plane is the plane of operation.

G18: XZ-plane, tool in Y-axis

G19: YZ-plane, tool in X-axis

Refer to G19 using an angular head for programming an angular head.

Changing the plane of operation

When a new plane is selected, thus either G18 or G19 activated, the length compensation in the Z-axis is cancelled and activated in the tool axis related to the selected plane.

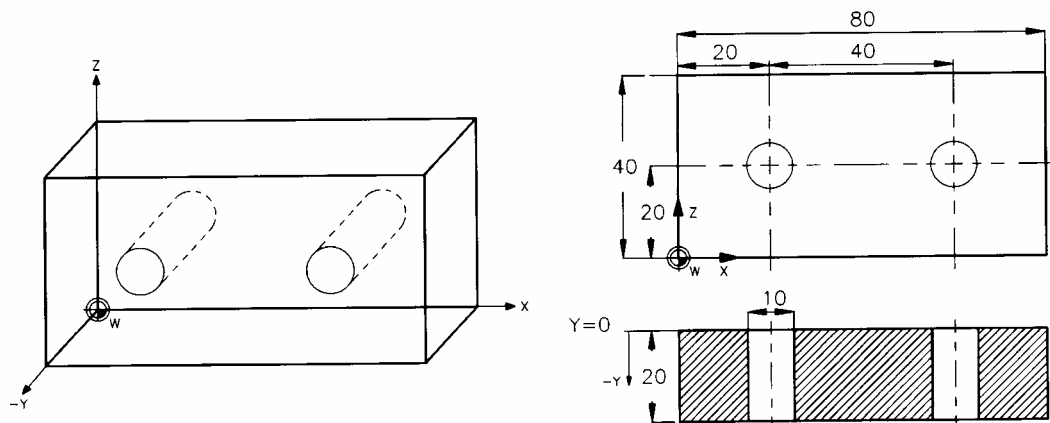
Cancellation

The G17 function is cancelled by activating another machining plane, using either the G18 or G19 functions. The G17 function is not cancelled by CLEAR CONTROL or by softkey CANCEL PROGRAM.

Tool offsets

Tool dimensions stored in the Tool Memory are independent of the selected plane.

Example



```
N9001
N1 G17
N2 T1 M6
N3 G0 X20 Y20 Z1 F400 S1600 M3
```

```
N4 G1 Z-23.5
N5 G0 X60 Z1
```

```
N6 G1 Z-23.5
N7 G0 Z200
N8 M30
```

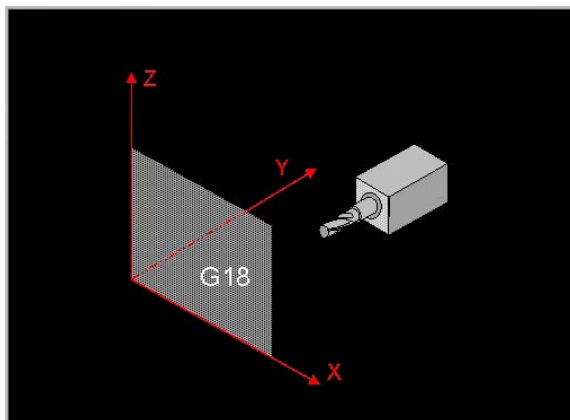
Activate G17 main plane.
Load tool T1 and its offsets. Drill diameter is 10 mm.
Move tool rapidly (G0) to programmed position. Set feedrate to 400 mm/min. Make spindle. Rotate clockwise (M3) at 1600 rev/min.
Feed tool to programmed depth.
Retract tool to Z1 and then move the tool rapidly to X60. The CNC's positioning logic ensues. That the tool does not collide with the workpiece, because the tool is first moved along the Z-axis before moving along the X-axis.
Feed tool to programmed depth.
Retract tool to Z200
End of program.

5.12 G18 Mainplane XZ, tool Y

The main spindle of the machine tool determines the position of the tool axis. With G18 is defined that the tool axis is the Y-axis and the main plane for milling operations the XZ-plane.

Format

G18



Notes and usage

Modality

This function is modal with G17 und G19.

Modal words

F, F1=, F3=, F4=, S, T, T1=, T2=, T3=, M, H, Ennn

It is advised to program this function in a separate block without modal parameters.

Default plane

When switching on the machine or after a CLEAR CONTROL the machine activates automatically due to MC11 (0=G17, 1=G18, 2=G19) a PLANE.

The last selected PLANE is active when the machine is switch on normally.

Operations in the plane

Calculations for radius compensation, the geometry (G64), polar coordinate, milling cycles; the pocket cycle, etc. are performed in the current plane. Thus when G18 is active the XZ-plane.

Operations in the tool axis

Toollength compensation and the fixed cycles for hole operations use the current tool axis. Thus when G18 is active the Y-axis.

Angular head

When an angular head is fitted, the axis configuration of the machine tool remains unchanged. So the tool can be in either the Z- or X-axis.

With the function G17 or G19 is programmed in which axis the tool is standing and which plane is the plane of operation.

G17: XY-plane, tool in Z-axis

G19: YZ-plane, tool in X-axis

Refer to G19 using an angular head for programming an angular head.

Changing the plane of operation

When a new plane is selected, thus either G17 or G19 activated, the length compensation in the Y-axis is cancelled and activated in the tool axis related to the selected plane.

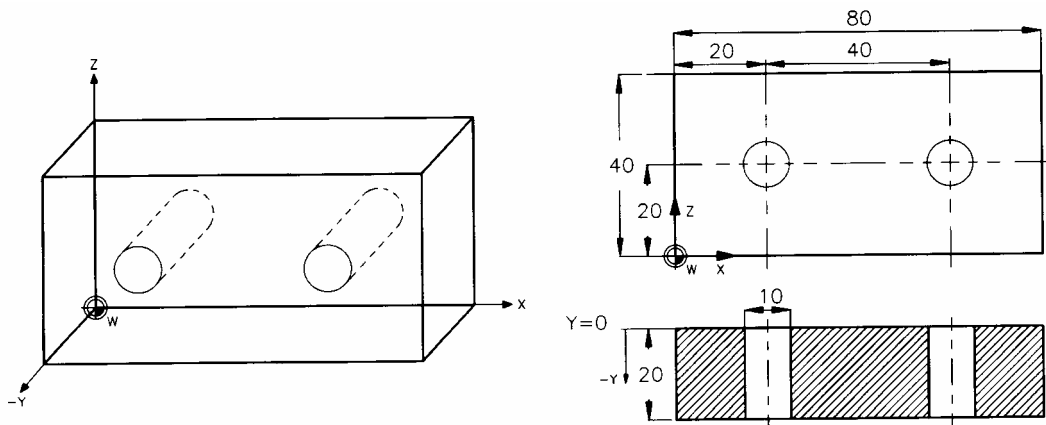
Cancellation

The G18 function is cancelled by activating another machining plane, using either the G17 or G19 functions. The G18 function is not cancelled by CLEAR CONTROL or by softkey CANCEL PROGRAM.

Tool offsets

Tool dimensions stored in the Tool Memory are independent of the selected plane.

Example



N9002

N1 G18

N2 T2 M6

N3 G0 X20 Y1 Z20 F400 S1600 M3

N4 G1 Y-23.5

N5 G0 X60 Y1

N6 G1 Y-23.5

N7 G0 Y200 M30

Make XZ-plane (G18) active.

Load tool T2 and its offsets. Drill diameter is 10 mm.

Move tool rapidly (G0) to the programmed position. Set the feedrate to 400 mm/min and

Make spindle rotate clockwise (M3) at 1000 rev/min.

Feed tool to depth.

Retract tool to Y1 and then move tool rapidly to X60. The CNC's positioning logic ensures that

The tool does not collide with the workpiece, because the tool is first moved along the Y-axis, before moving along the X-axis.

Feed tool to depth.

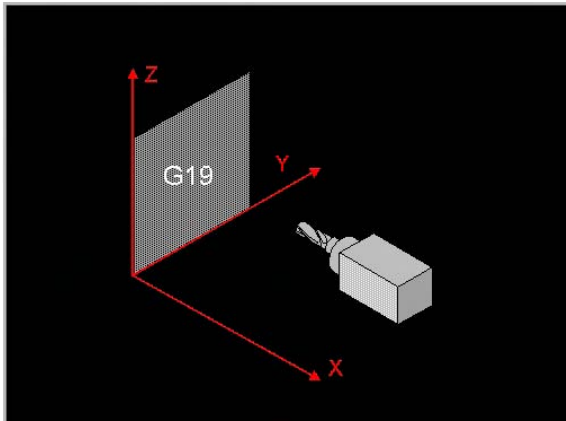
Retract tool to Y200 and end of program.

5.13 G19 Mainplane YZ, tool X

The main spindle of the machine tool determines the position of the tool axis. With G19 is defined that the tool axis is the X-axis and the main plane for milling operations the YZ-plane.

Format

G19



G Mainplane YZ, tool X

Notes and usage

Modality

This function is modal with G17 und G18.

Default plane

When switching on the machine or after a CLEAR CONTROL the machine activates automatically due to MC11 (0=G17, 1=G18, 2=G19) a PLANE.

The last selected PLANE is active when the machine is switch on normally.

Operations in the plane

Calculations for radius compensation, the geometry (G64), polar coordinate, milling cycles; the pocket cycle, etc. are performed in the current plane. Thus when G19 is active the YZ-plane.

Operations in the tool axis

Toollength compensation and the fixed cycles for hole operations use the current tool axis. Thus when G19 is active the X-axis.

Angular head

When an angular head is fitted, the axis configuration of the machine tool remains unchanged. So the tool can be in either the Z- or Y-axis.

With the function G17 or G18 is programmed in which axis the tool is standing and which plane is the plane of operation.

G17: XY-plane, tool in Z-axis

G18: XZ-plane, tool in Y-axis

G19 YZ-plane, tool in negative or positive X-axis

Changing the plane of operation

When a new plane is selected, thus either G17 or G18 activated, the length compensation in the X-axis is cancelled and activated in the tool axis related to the selected plane.

Cancellation

The G19 function is cancelled by activating another machining plane, using either the G17 or G18 functions. The G19 function is not cancelled by CLEAR CONTROL or by softkey CANCEL PROGRAM.

Tool offsets

Tool dimensions stored in the Tool Memory are independent of the selected plane.

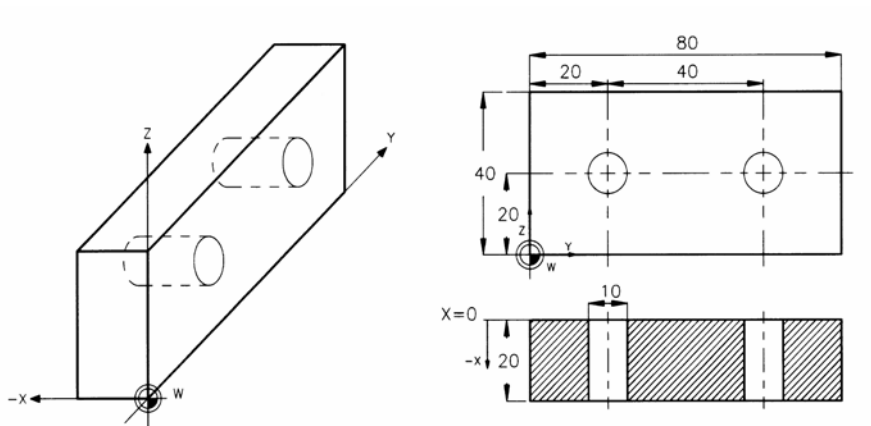
Using an angular head

When an angular head is used, its dimensions must be programmed. Either a zero point shift (G92 or G93) or a stored zero offset (G54 - G59) can be used for this purpose. The use of stored zero offsets is recommended because the partprogram remains independent of the dimensions of the angular head.

Tool in + or - direction of X-axis

Especially with an angular head in the X-axis the tool can be in the positive (+) or negative (-) direction of the axis. The functions G66 and G67 are available to indicate in which direction the tool is standing and allows the partprogrammer to look always in the same way at the plane of operation. Refer to G66/G67 for using these functions.

Example



N9003
N1 G19
N2 T3 M6
N3 G0 X1 Y20 Z20 F400 S1600 M3

N5 G0 X1 Y60

N6 G1 X-23.5
N7 G0 X200 M30

Make YZ-plane (G19) active.
Select tool T3 and its offsets. Drill diameter is 10 mm.
Move tool rapidly (G0) to programmed position. Set the feedrate to 400 mm/min and make Spindle rotate clockwise (M3) at 1000 rev/min.
N4 G1 X-23.5 Feed tool to depth.
Retract tool to X1 and then move tool rapidly to Y60. The CNC's positioning logic ensures that The tool does not collide with the workpiece, because the tool is first moved along the X-axis, before moving along the Y-axis.
Feed tool to depth.
Retract tool to X200 and end of program.

5.14 G22 Macro call

To execute a subprogram with standard operations.

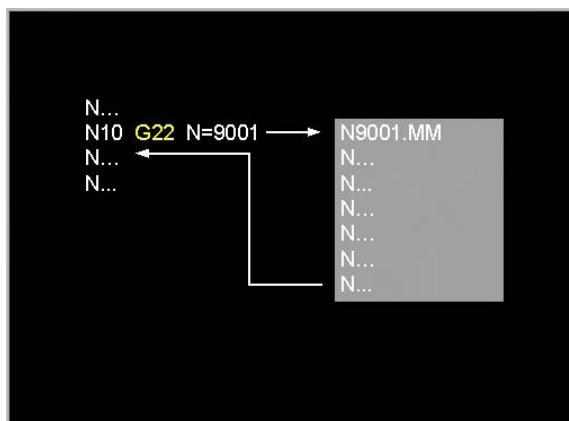
Format

To call a subprogram.

G22 N=... {E...=}

To activate a subprogram on the condition that E...>0

G22 E... N=... {E...=}



G Macro call
E Parameter definition
N= Macro number

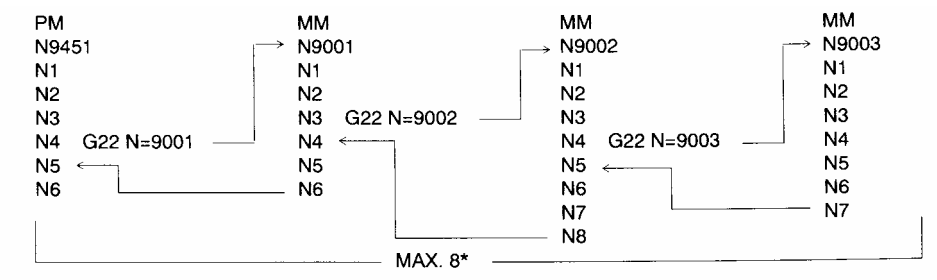
Notes and usage

Activation

A subprogram is completely executed when it is called from a main program or from another subprogram.

Nesting of subprograms

When a subprogram calls another subprogram, the subprogram, which is activated, is referred to as a 'nested' subprogram. At the end of a nested subprogram the calling subprogram continues. A maximum of eight 'nested' programs can be used.



E-parameters

A subprogram may contain E-parameters, which are variables whose values are stored in a separate CNC memory. Subprograms can therefore be written which have a general application. When the dimensions of a component are known, only the E-parameter values need to be altered, not the program.

E-parameters can get their value in the main program or subprogram, via the operator's panel, or by reading-in the parameter memory.

Arithmetical calculations with parametric values are allowed in programs and subprograms. The same parameter can be used by different (sub) programs.

Refer to the special appendix about E-parameters at the end of this manual for more details of programming with E-parameters.

Number of parameter definitions

In a block with a macro call up to 10 parameters can get their value. If more parameters are used, extra lines before the macro call are necessary.

Evaluation of defined parameters

Any value or arithmetical expression can be assigned to a parameter in a G22-block.

Parameters programmed in the G22 block are evaluated and calculated before the execution of the macro.

Continuation after the macro call

Once the macro is executed, the program continues with the block after the G22 in which the macro was called.

Conditional macro call (E)

The value of the E-word is used for dictating if a conditional macro call must be performed.

If the value of E...>0, the macro call is performed.

After the call the program continues with the block after the G22. Parameter E... is not influenced by the macro call.

If the value of E...<=0, the macro is not called. The program continues with the block after the G22.

Examples

Example 1 A macro call.

N100 G22 N=9100 E1=24 E2=3

Execute subprogram N9100 with parameters E1=24 and E2=3.

Example 2 Conditional macro call.

N150 G22 E60 N=9100

Execute subprogram N9100 when the value of E60 > 0.

Example 3 Macro without E-parameters.

Subprogram for drilling two holes:

N9001

N1 G91

N2 G1 Z-16 M8

N3 G0 Z16 M9

N4 X20

N5 G1 Z-16 M8

N6 G0 Z16 M9

N7 G90

Activate incremental programming.

Switch coolant ON. Move tool with a feedrate in negative direction.

Retract tool. Switch coolant OFF.

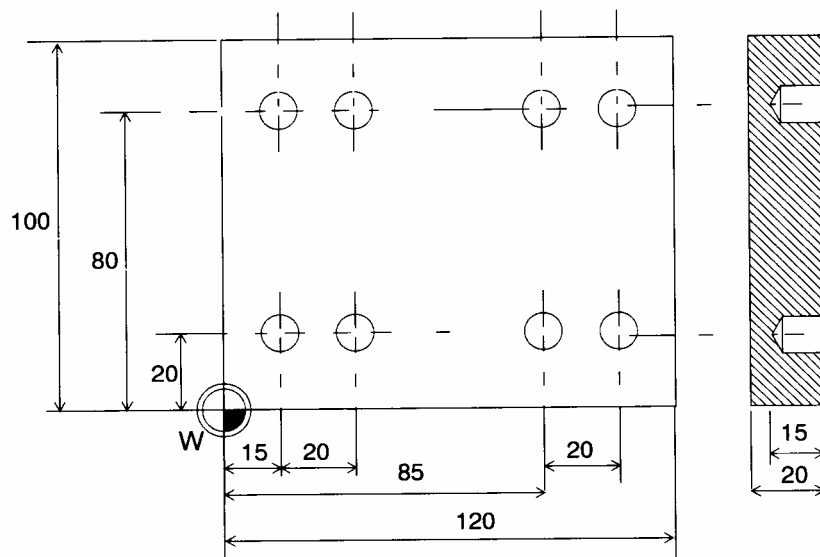
Move tool 20 mm along X-axis to second start position.

Switch coolant ON. Feed tool 15 mm into workpiece.

Retract tool. Switch coolant OFF.

Re-activate absolute programming.

Example 4 Main program for drilling four pairs of holes:



Macro program

N9001

N1 G91

N2 G1 Z-15

N3 G0 Z16

N4 G90

Main program

N45 T1 M6

N50 F400 S1600 M3

N55 G0 X15 Y20 Z1

N60 G22 N=9001

N65 G0 X85

N70 G22 N=9001

N75 G0 X85 Y80

N80 G22 N=9001

N85 G0 X15

N90 G22 N=9001

Load tool T1 and use its offsets. Drill diameter is 10 mm.

Make spindle rotate clockwise at 1600 rev/min. Feedrate at 400 mm/min

Move tool to first drilling position and 1 mm off top surface.

Activate subprogram

Move tool to second drilling position.

Activate subprogram

Move tool to third drilling position.

Activate subprogram

Move tool to fourth drilling position.

Activate subprogram

Programming instruction

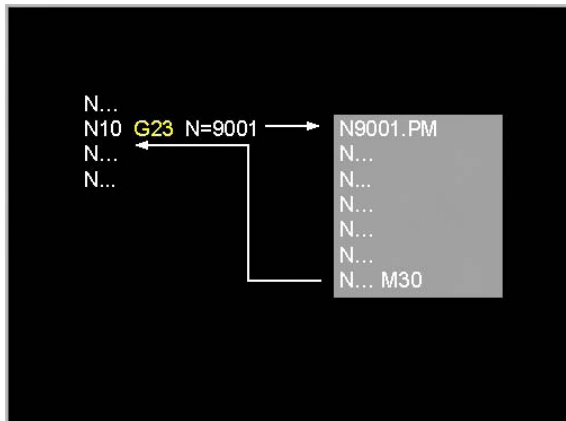
G0 is not really necessary in N65, N75 and N85 blocks, because G0 is programmed in the last block of the subprogram. G0 has been programmed in the latter block, because then you need not know how exactly the subprogram ends in order to understand the main program.

5.15 G23 Main program call

To call a partprogram from a main program.

Format

G23 N=... {N5=...}



G Main program call
N= Program number
N5= Directory

Notes and usage

The program to be called is defined by a program number (N) and possibly with path (N5=)

Definition of the path (N5=)

In the **SP**-version the total length of the path (N5=) and program number (N) has a maximum of 75 characters. In the **DP**-version this is a maximum of 115 characters.

In the **SP**-version programs only can be called via NFS (Network File System: See Technical Manual). In the **DP**-version programs can be called via the Windows network.

The definition of the path of programs in the CNC is:

G23 N1007	Program N1007 is called from the work directory. Mostly D:\work.
G23 N1007 N5= "test1\"	Program N1007 is called from the sub-directory "test1" from the work directory. Mostly D:\work
G23 N1007 N5= "\test1\"	Starting with \ means calling program N1007 from the subdirectory "test1" on the root directory of the hard disk. Mostly the root directory is D:. Only local drives except C: are allowed

The definition of the path of programs on a network (only **DP**-version) is:

G23 N1007 N5= "\\server1\test1\"	Starting with \\ means calling program N1007 via a network from directory \\server1\test1 on an external hard disk.
G23 N1007 N5= "S:\test1\"	Direct calling program N1007 via a network from directory "test1" on the drive S:. Local drives [C: D: {E:} {F:}] are not allowed.

Example:

Program example	Description
	Work directory is D:\WORK\
N10 G23 N1007 N5="test1\"	File from D:\WORK\TEST1\ is called
N20 G23 N1007 N5="test2\"	File from D:\TEST2\ is called
N30 G23 N1007 N5=""	File from D:\ is called
N40 G23 N1007 N5="c:\test3\"	Error message
N50 G23 N1007 N5="z:\test4\"	SP: File from NFS-directory Z:\TEST4\ is called.

	DP and WinShape: File from Windows network Z:\TEST4\ is called
N60 G23 N1007 N5="//server1/test5\"	SP : Error message. DP and WinShape: File from Windows network \\SERVER1\TEST5\ is called

Program size

Programs smaller than 100 Kbytes will be stored in the work-memory and executed as a normal G23 call.

Programs bigger than 100 Kbytes cannot be stored in the work-memory. They will be separated automatically and invisible, in a lot smaller partprograms. These partprograms will be executed automatically (CAD MODE).

Restrictions

A called partprogram cannot contain a G23-function; partprograms cannot be "nested" within one another.

A subprogram (macro) must not contain the G23-function.

Programs bigger than 100 Kbytes may not have jump instructions.

Continuation after the program call

Once the called program is executed, the main program continues with the block after the G23 in which the program was called.

Termination of a called program

When the execution of a called partprogram will be stopped with Intervention or softkey <Cancel program>, a jump to the mainprogram start will be done.

Example

Programming example	Description
N9990	Programm-Nummer
N10 G23 N=988	Program N988 is called
N20 G23 N=989	Program N989 is called
N30 M30	
N988	Program N988
N1	
N	
N200 M30	Jumping back to main program N9990

5.16 G25/G26 Enable/Disable feed- and/or speed-override

To enable or disable the feed- and/or speed-override, in order to control programmed feed and speed movements. If the feed- or speed-override is disabled, the feed- or speed-override is fixed to 100%.

Format

To enable feed- and speed-override:

G25

To disable feed override (F=100%):

G26 I2=1 or without I2

To disable speed override (S=100%):

G26 I2=2

To disable feed- and speed-override (F and S=100%):

G26 I2=3



Notes and usage

Modality

G25 and G26 are modal function.

Default mode

The CNC system automatically activates G25 at the start of a partprogram.

Cancellation

The G26-function is cancelled by G25 or with softkey CLEAR CONTROL or softkey CANCEL PROGRAM or M30.

Example

N66 G26 I2=1

Feed override switched off, thus F fixed to 100%.

N67 G26 I2=2

Speed override switched off, thus S fixed to 100%.

N68 G26 I2=3

Feed- and speed-override switched off, thus F and S fixed to 100%.

N70 G25

Feed override switched on.

5.17 G27/G28 Positioning functions

1. For indicating with feed movements (G1, G2/G3, G6) when the next movement starts and if a stop between the movements should occur.
 2. For indicating with a rapid movement (G0) if a stop between the movements should occur or must be avoided. Parameter I4=.
 3. For switching off and on the positioning logic with G0 movements. Parameter I5=.
 4. Controlling contour tolerance. Parameter I7=.
- Note: The Look Ahead Feed-Function (LAF) function controls the path with the highest possible accuracy (<10 µm). However, high accuracy is at the expense of path speed, because the control has to decelerate at each edge to guarantee the contour tolerance. In certain cases (e.g. when roughing), speed is more important than accuracy and this function is not wanted.
5. Affecting of the acceleration. Parameter I6=

Format

To activate:

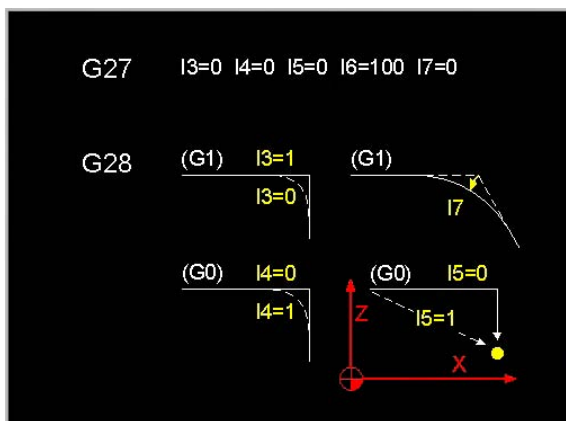
G28 {I3=...} {I4=...} {I5=...} {I6=...} {I7=...}

To cancel each possibility separately

G28 {I3=0} {I4=0} {I5=0} {I6=100} {I7=0}

To cancel all possibilities (default setting):

G27



G Positioning functions
 I3= Feed movement 0=inpos, 1=inpod
 I4= Rapid movement 0=inpod, 1=inpos
 I5= Position logic: 0=with, 1=without
 I6= Reduction acceleration/jerk [%]
 I7= Contour tolerance

Notes and usage

Modality

G27 and G28 are modal function.

Movements

Movements with inposition

A movement with inposition means that the next movement starts once all programmed axes have reached their programmed position. A stop occurs between the movements.

Movements without inposition

A movement without inposition means, that the next movement starts as soon as the interpolator of the CNC has reached the commanded position. No deceleration or axes lag is taken into account. There is no stop between the movements, so the program is executed faster, but with feed movements rounding of corners occur.

Feed movements

Feed movements (G1, G2/G3)

Parameter I3 controls the point at which the next programmed movement starts after a feed movement.

G28 I3=0

The feed movement is executed WITHOUT INPOSITION. There is no stop between the movements and therefore corners are rounded; machining quality is good.
I3=0 is the default setting with feed movements.

G28 I3=1

The feed movement is executed WITH INPOSITION. A stop occurs between the movements. Corners are sharp, but machining quality is poor.

Rapid traverse movements**Rapid traverse movements (G0)**

Parameter I4 controls the point at which the next programmed movement starts after a rapid movement (G0).

G28 I4=0

The rapid movement is executed WITH INPOSITION. A stop occurs between the movements.

I4=0 is the default setting with rapid movements.

G28 I4=1

The rapid movement is executed WITHOUT INPOSITION. There is no stop between the movements.

Positioning logic**Switching on/off the positioning logic with G0**

Parameter I5 indicates if the positioning logic in a G0-block should be executed or switched off. Refer to the function G0 for a description of the positioning logic.

G28 I5=0

G0 is executed with positioning logic.

I5=0 is the default setting.

G28 I5=1

Positioning logic is not active with a G0 movement

Note: The positing logic with a G79 block cannot be switched off.

Programmable acceleration and jerk reduction

The acceleration and jerk per axes are normally determinate by machine constants (MC3*04 and MC3*05)). This acceleration and jerk will be multiplied with the acceleration and jerk reduction. Value between 5 and 100 can be given (5 is a very small acceleration, 100 is normal).

This reduction is active for G0, G1, G2 and G3, which are executed with LAF.

G28 I6=... (5 to 100 %)

G28 I6=100 Setting back to normal value (100 %)

Programmable contour accuracy (rapid and feed)**G28 I7=**

The permissible contour accuracy, in mm (0 – 10.000 mm).

If I7= is not programmed, the numerical value established via a machine constant (MC765) is used as maximum difference.

G28

When carrying out the feed or rapid movement, the contour accuracy programmed with I7=.. must be taken into account.

The feed is automatically reduced by the CNC to the maximum feed at which the corner can be executed. The programmed difference is not exceeded.

Notes:

1. The programmable contour accuracy cannot be used for corners connected with splines.
2. The input I7=0 is turned off (MC765 is taken into account).

Cancellation parameters

Cancellation all parameters

All G28 parameters are reset to their default values by programming G27 or by performing the CLEAR CONTROL operation or softkey CANCEL PROGRAMM or M30.

G27 results in G28 I3=0 I4=0 I5=0 I6=100 I7=0

Cancellation of each individual parameter

Each parameter of the G28 function can be cancelled separately by programming the default setting. The parameters do not influence each other.

Overview:

- | | |
|---|------------|
| 1. G28 without parameter | |
| G1, G2, G3 without In-Position | G28 |
| 2. Movement with feed | |
| G2, G3 without In-Position (initial setting) | G28 I3=0 |
| G1, G2, G3 with In-Position | G28 I3=1 |
| 3. Rapid traverse movements G0 | |
| G0 with In-Position (initial setting) | G28 I4=0 |
| G0 without In-Position | G28 I4=1 |
| 4. Positioning logic with G0 | |
| G0 with positioning logic (initial setting) | G28 I5=0 |
| G0 with positioning logic | G28 I5=1 |
| 5. Acceleration and jerk reduction | |
| G0, G1, G2, G3. | |
| -Acceleration and jerk per axes | |
| (MC3*04 and MC3*05)) | G28 I6=100 |
| -Acceleration reduction | G28 I6=... |
| I6= 5 to 100 % | |
| 6. Movements with programmable contour accuracy | |
| G0, G1, G2, G3 | |
| -Contour accuracy (MC765) | |
| -Programmable contour accuracy | |
| I7=... (0-10.000 µm) | G28 I7=... |

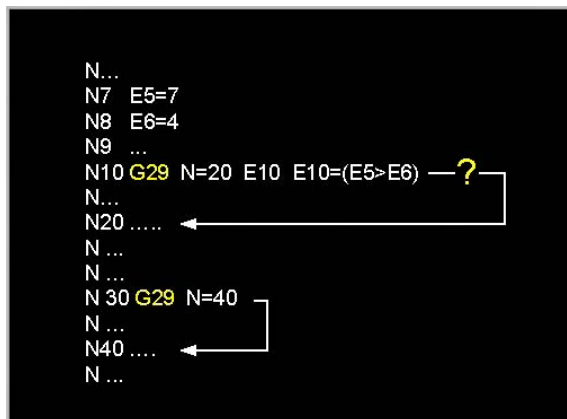
5.18 G29 Jump function

To jump to a different section of a partprogram (or subprogram) if a parameter is >0.

Other jump conditions like =, <>, >, >=, <, <=, can be programmed when using a relational expression together with the G29-function.

Format

G29 {E...} N=... {K...} {I...}



G Jump function
I Search direction
K Jump decrement
E Jump condition: E > 0
E*** Parameter definition
N= Jump to blocknumber

Notes and usage

Block number for jump (N=)

This word specifies the block number of the block to jump to. The block must be in the same partprogram or subprogram.

When there are block numbers with the same number then the first following block with the number is jumped to or a error is shown.

Jump direction

A jump can be performed in forward or backward direction in a (sub) program. With I=1 or I=0 the jump is performed forwards. If I=-1 or nothing the jump is performed backwards to the top of the program and the forwards.

Jump condition (E)

The value of the E-word is used for dictating if a conditional jump must be performed.

If the value of E...>0, a jump is performed.

If the value of E...<=0, the jump is not performed.

Jump decrement (K)

The K-word allows the value of the E-word to be recalculated and used as the condition for jumping.

If the K-word is not programmed, the E-word's value is reduced by 1 every time a jump is made.

If K0 is programmed, the value of the E-word is not reduced.

If K>0, its value is used to reduce the E-word's value. If eg. K5 were programmed, 5 would be subtracted from the E-word's value after every jump.

If K<-.5 an error message is displayed.

Defining e-parameters in a G29-block

In a G29-block parameters can be defined and calculated. The sequence of executing the block is:

1. load the parameters
2. perform the jump, if the condition is fulfilled

Unconditional jump

An unconditional jump can be programmed without jump condition.

Eg. G29 N=...

Relational expressions

When using relational expressions (see the appendix on E-parameters at the end of this manual), the programming facilities of the conditional jump are substantially expanded.

The relational expression sets the parameter for the jump condition to 0 or 1. The jump is executed as usual.

To keep a program as readable as possible, it is advised to program the relational expression in the same block as the G29. However, the relational expression can be programmed also in a block before the G29, as only the set parameter for the jump condition is used by the G29.

Eg. N.. G29 E1=E2>E3 E1 N=400

This block means:

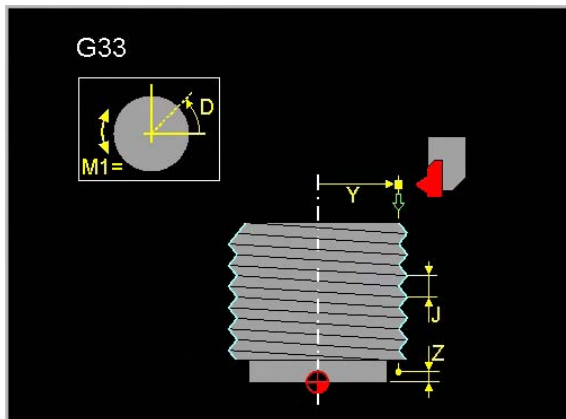
If the value of E2 is greater than the value of E3, parameter E1 is set =1 and on this setting the jump to N400 is performed.

Example

:	
N50 E2=3	Set initial value of the parameter E2 to 3.
N51	
:	
N100 G29 N=51	Jump to block number 51
:	
N100 G29 E2 N=51	When E2 > 0, jump to block number 51 and then continue to execute the program blocks in sequential order till block N100.

At each jump parameter E2 is decremented automatically by 1. Therefore, after 3 loops, parameter E2 is equal to zero and no more jumps executed. The program continues after block N100 in the sequential order.

5.19 G33 Basic Threadcutting movement

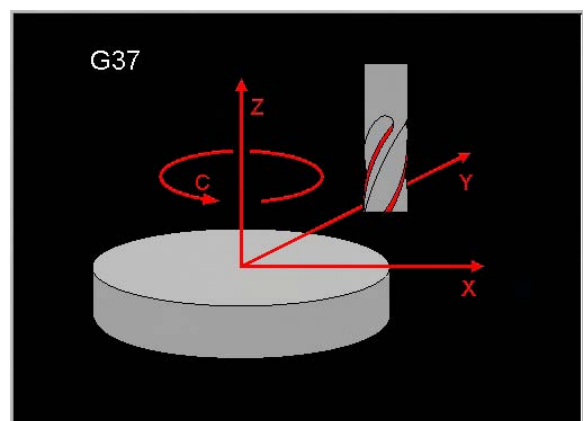
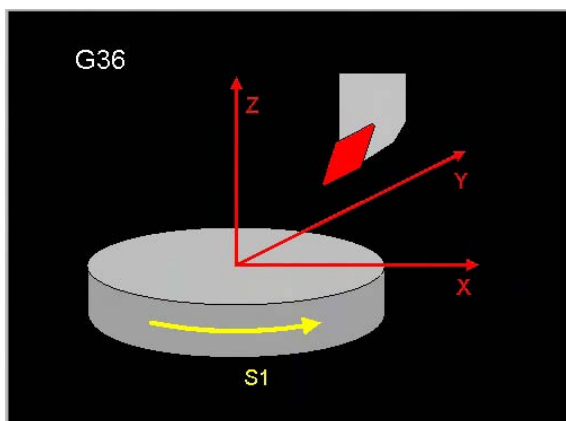


```

G   Single threadcutting movement
X   Endpoint coordinate
Y   Endpoint coordinate
Z   Endpoint coordinate
J   Pitch
D   Start angle threadcutting
?90= Endpoint abs. (X,Y,Z..)
?91= Endpoint incr. (X,Y,Z..)
  
```

Refer to Chapter "Turning mode".

5.20 G36/G37 Activate/ Deactivate turning mode



Refer to Chapter "Turning mode".

5.21 G39 Activate/Deactivate tool offset

Programmed contours can be changed by an offset

Format

Activate offset:

G39 {R...} {L...}

R: Tool radius offset

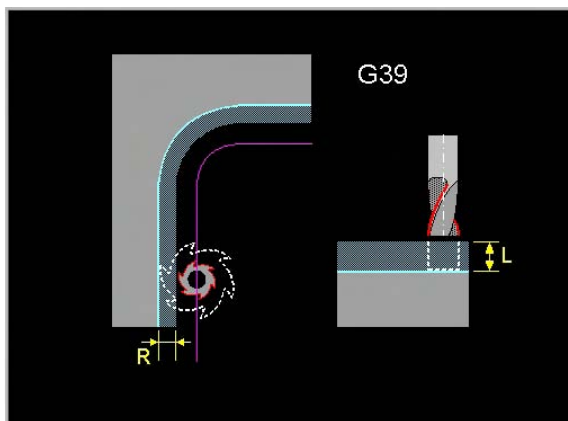
L: Tool length offset

Deactivate tool length offset:

G39 L0

Deactivate tool radius offset:

G39 R0



```
G  Activate tool offset
L  Toollength offset
R  Toolradius offset
```

Notes and usage

Tool length offset:

The tool length offset operates into the direction of the tool axis. Tool length offset changes will become effective with the next feed movement.

Tool radius offset:

The tool radius offset operates in the machining plane, but is only effective with active cutter radius compensation.

If the cutter radius compensation is inactive, tool radius offset changes will become effective after the cutter radius compensation (G41/G42, G43/G44) has been activated.

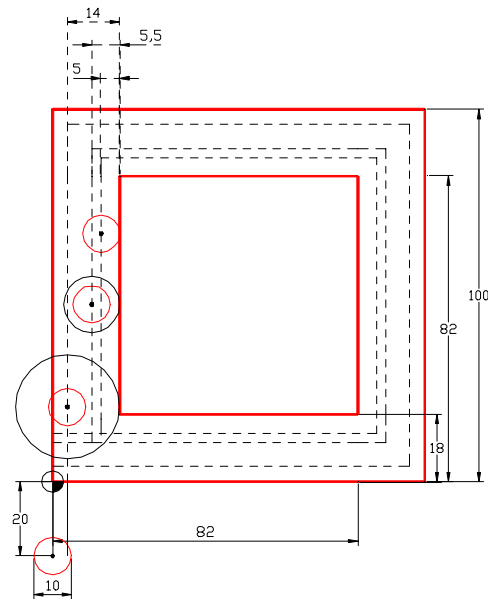
If the cutter radius compensation is active, tool radius offset changes will be corrected in the next movement block linearly over the entire path.

Offset programming is maintained after a tool change (M6, M66) or change of plane (G17, G18, G19).

Note: There will be a radius offset override when the following functions are activated: G6, G83-G89, G141, G182. The length offset remains effective. Offset programming should be deactivated before these functions.

Example

Rectangular milling by roughing (2x) and finishing (1x)



N39001

N1 G98 X-10 Y-10 Z10 I120 J120 K-60 Define graphic window

N2 G99 X0 Y0 Z0 I100 J100 K-40 Define material

N3 T1 M6

N4 **G39 L0 R9** Change tool (cutter diameter 10 mm)Activate tool radius offset. (Cutter radius for radius compensation is $(5+9 =) 14$ mm)

N5 F500 S1000 M3

Activate feed and spindle speed

N6 G0 X0 Y-20 Z5

Approach starting position

N7 G1 Z-10

Moving to depth

N8 G43 X18

Approach contour with radius compensation

N9 G41 Y82

Initial roughing of the rectangle. Offset is 9 mm.

N10 X82

N11 Y18

N12 X0

N13 G40

Turn off radius compensation.

N14 **G39 R0.5**Change tool radius offset. (Cutter radius for radius compensation is $(5+0.5 =) 5.5$ mm)

N15 G14 N1=8 N2=13

Repeat rectangle (second roughing operation). Offset for finishing is 0.5 mm

N16 **G39 R0**

Change tool radius offset. (Cutter radius for radius compensation is 5 mm)

N17 G14 N1=8 N2=13

Finish the rectangle.

N18 G0 Z10

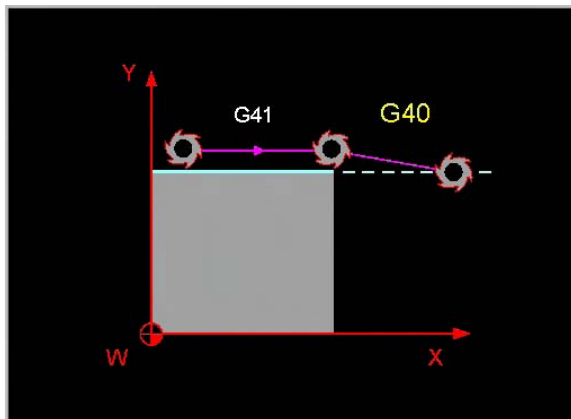
Approach clearance distance

N19 M30

Program end

5.22 G40 Cancel tool radius compensation

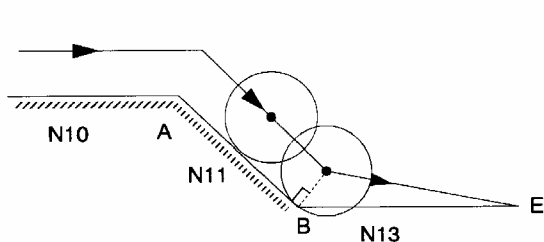
To cancel radius compensation. The tool now moves along the programmed path on the workpiece.



G Cancel tool radius compensation

Format

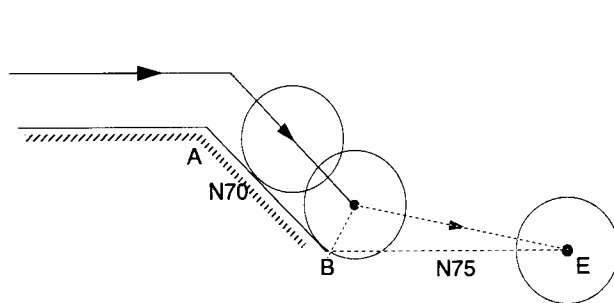
General: G40 {axis coordinates}



```
N... G41
:
N11 G1 Xb Yb
N12 G40
N13 Xe Ye
```

G40 {without movement}

The radius compensation LEFT is active from point A to point B. At point B, radius compensation is cancelled and programmed movements refer to the tool point.



```
N50 G41
:
N70 G1 Xb
N75 G40 Xe
:
```

G40 {with movement}

Radius compensation LEFT is active from point A to point B. At point B, radius compensation is cancelled and programmed movements refer to the tool point.

Notes and usage**Modality**

This function is modal with G41, G42, G43 and G44.

Default mode

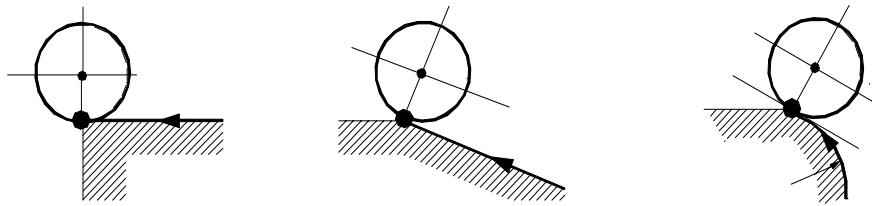
G40 is made active automatically when the CNC system is switched on, M30 and softkey CLEAR CONTROL and CANCEL PROGRAM operations are performed.

Axes coordinates

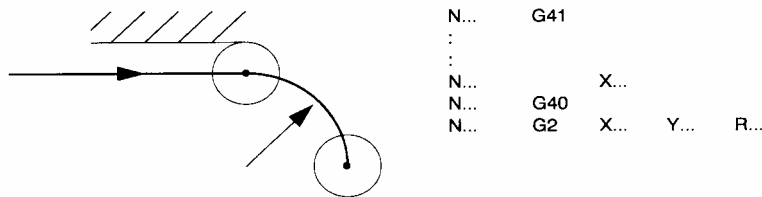
It is advised to program G40 in a separate block without axis coordinates.

(a) G40 without axis coordinates.

In this case, the previous line or circle will be cut completely. The centre of the tool will be moved to a position perpendicular to the contour in the previous endpoint. The compensation is switch off in the next movement.



G40 followed by a linear movement



G40 followed by a circular movement

If a circular movement follows the G40-block, an arc with the programmed radius is inserted between the corrected endpoint of the G40-block and the programmed tooltip position in the block with the circle

(b) G40 with axis coordinates.

The start point of the movement is calculated with full compensation. During the movement the compensation is switched off and the endpoint is without compensation. This way of programming can be used when the compensation can be switched off without damaging a contour.

(c) Tangential exit.

It is also possible to leave a contour with G62.
N.. G62 X.. Z.. R.. (F)

Lifting the tool from the plane of operation

The tool can be lifted from the plane of operation with

(a) A linear movement

If the plane of operation is e.g. the XY-plane and programming a linear movement in YZ lifts the tool, the tool correction in the third axis (X) is cancelled too. So a movement takes place in three axes simultaneously.

(b) A circular movement

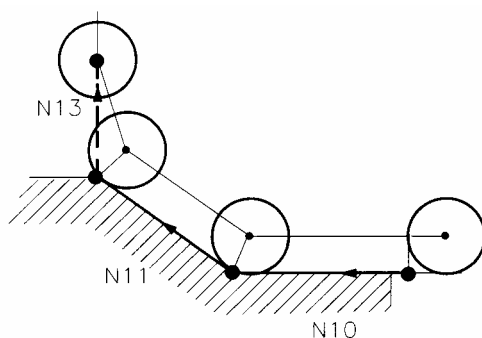
If the plane of operation is e.g. the XY-plane and programming a circular movement in YZ lifts the tool, the tool correction in the third axis (X) is not cancelled. So only the circular movement in the YZ-plane is executed.

Note: The start position of the circle is in the Y-axis the corrected position and in the tool axis (Z) the depth of operation.

Note: Programming sub sequential identical axes positions in G43 and G40 may cause a positioning error in the axis concerned over the toolradius. This can be prevented by programming in G40 an axis position slightly different (e.g. 1 micron) from the previous position in G43.

Example: N100 G43 X10
 N101 G40 X10.001

Exampel



N9 G42
N10 G1 X..

activate radius compensation on right side of contour
move tool to programmed coordinates. Include tool radius into calculations.

N11 X... Y...
N12 G40
N13 G0 Y...

cancel radius compensation.
move tool from the previous compensated position to the uncompensated endpoint of this rapid movement.

5.23 G41/G42 Tool radius compensation (left/right)

To allow for workpiece dimensions to be programmed rather than the toolpath. The toolpath is automatically calculated by the CNC to be a path parallel to the programmed workpiece contour.

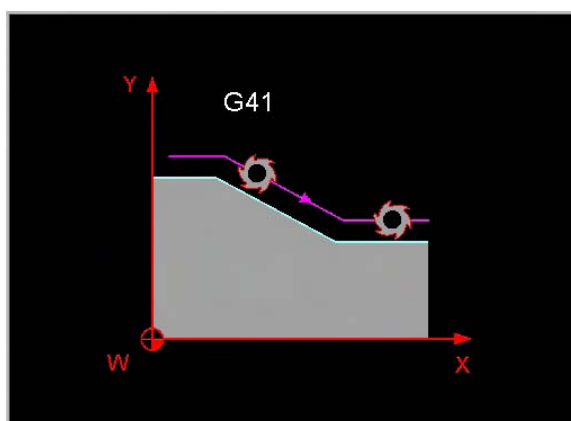
G41 activates radius compensation LEFT of the workpiece

G42 activates radius compensation RIGHT of the workpiece

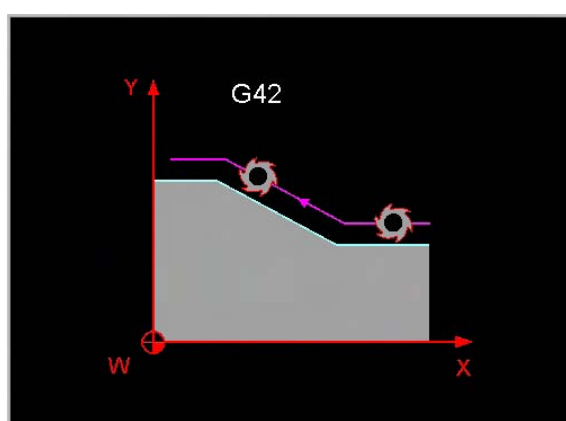
In both cases when looking in the same direction as the movements of the cutting tool.

Format

G41/G42 {axis coordinates}



G41



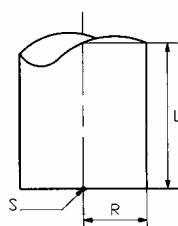
G42

Notes and usage

Modality

This function is modal with G41, G42, G43 and G44.

Position of tool point



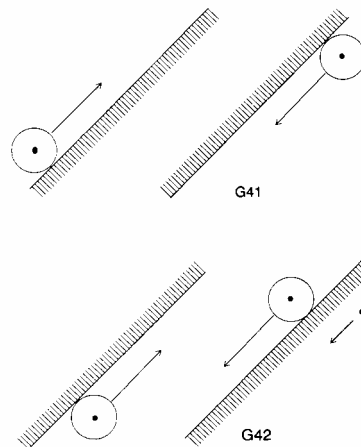
S = tool point specified by the tool dimensions L and R

Tool memory

The radius compensation function uses the tool radius from the tool memory.

Compensation left and right

When using radius compensation the CNC must know whether the tool is cutting at the left or right side of the workpiece. The function G41 or G42 is used for this purpose.



To decide which function must be programmed, it is necessary to look in the same direction as the movement of the cutting tool. If the tool moves on the left of the workpiece surface, G41 is used and on the right, G42.

This method assumes that a positive radius value is stored in the tool memory, when the program is executed.

However, if the stored radius value is negative, the following applies:

G41 and negative radius = G42 and positive radius

G42 and negative radius = G41 and positive radius

Refer to TOOL RADIUS CORRECTION for using negative radius values in the tool memory.

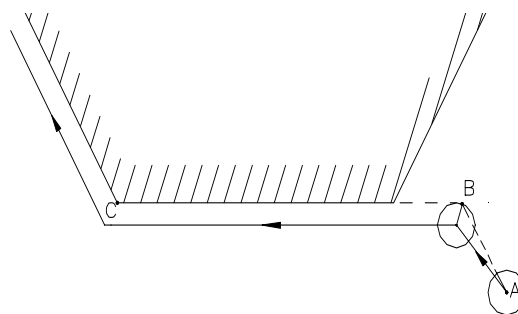
Starting radius compensation

There are three options to start radius compensations:

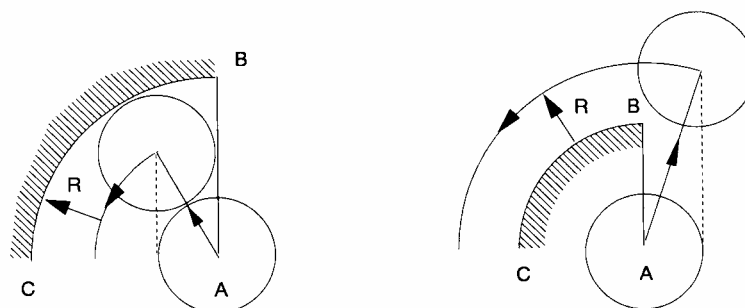
- 1) Direct with the functions G41/G42.
- 2) With the functions G43/G44. (See G43)
- 3) Or the function tangential approach. (See G61)

The partprogrammer must ensure that the tool does not collide with the workpiece, when radius compensation is being started. The start point must therefore be at a safe distance outside the workpiece.

When G41 or G42 is used, the intersection point between two related contour elements is calculated by the CNC and the tool moves to this point.

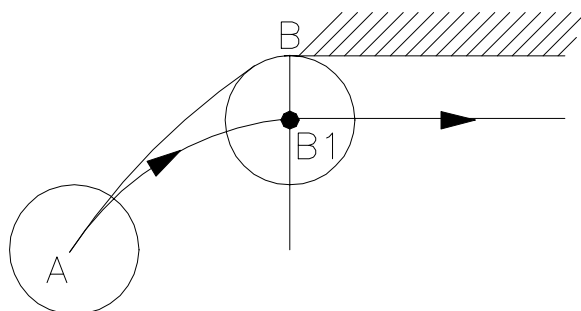


----Programmed path



Activate radius compensation G41/G42 with line to circle.

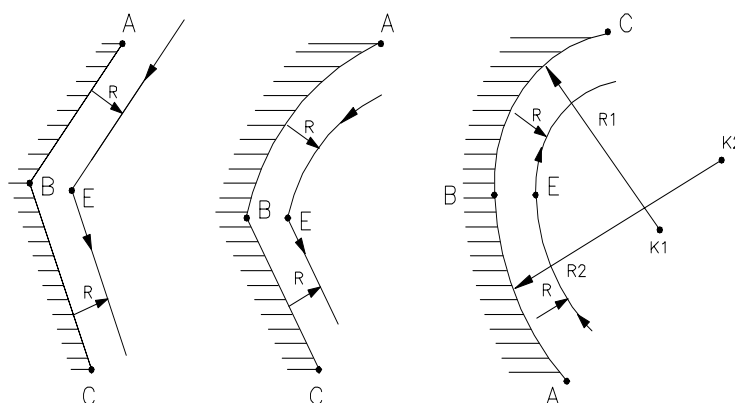
If radius compensation is activated during a circular movement (AB), the tool moves with a circular arc from the point the tool is standing (A) to the first calculated position (B1).



Activate radius compensation G41/G42 on a circle

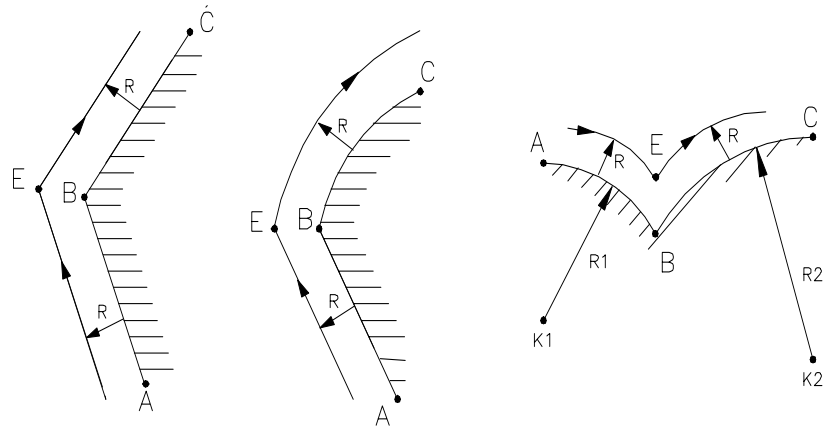
Internal contours

When radius compensation is used, the toolpath is always the same distance from the programmed contour, except at intersection points between contour elements. These points are calculated by the control automatically.



External contours

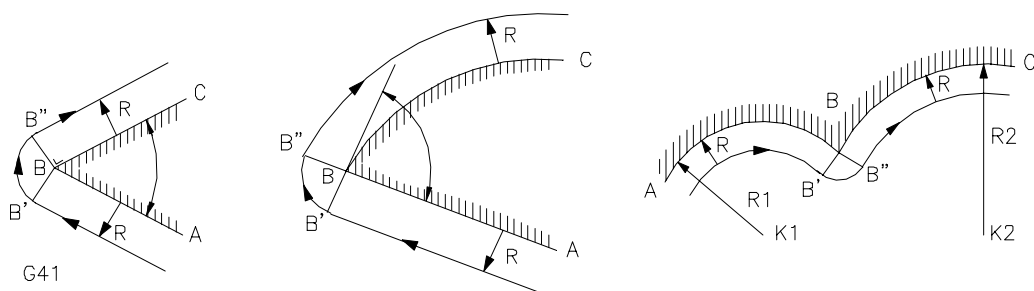
Intersection points between external contour elements are calculated and the tool moves to that position, whenever the angle between the elements is greater than a Machine Constant Value.



External contours with sharp corners

If the angle between two external contour elements is less than the Machine Constant Value (MC711), a circular movement between the two elements is generated by the CNC.

This circular movement is treated as part of the previous block. Therefore, if a SINGLE BLOCK operation is commanded, the tool stops after this circular movement.



Tool radius correction

In general a NC programming system calculates the toolpath taking into account the radius of a nominal tool. The radius compensation as described, allows to use a real tool for machining the part and to use a deviation on the radius of the nominal tool to let the control calculate the path of the actual tool.

A correction value on the tool radius including a sign is therefore stored in the tool memory.

" + Correction value": for an oversized cutter, thus with a radius greater than the radius of the nominal tool.

" - Correction value": for an undersized cutter

Movements

Programming either G41 or G42 does not result in a preparatory function for a motion (G0, G1, G2 or G3) also being activated. The last programmed function for a movement remains active.

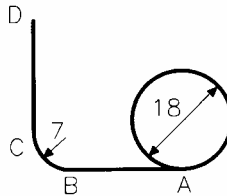
Programming errors

If the tool radius is too large, the workpiece might be damaged in some situations.

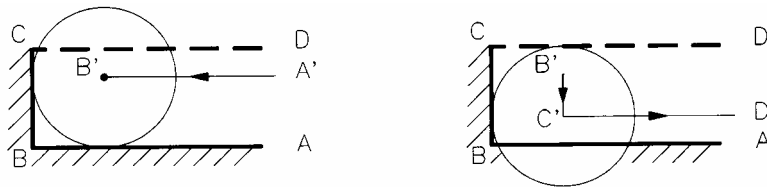
an error message is given in situation b, c and d. when G241 is activated,

- a The radius of the tool is equal to or larger than the workpiece radius. If this occurs an error message is generated.

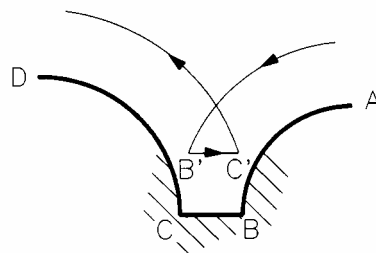
Note: the tool radius must be at least 0.001 mm (0.0001") smaller than the programmed radius.



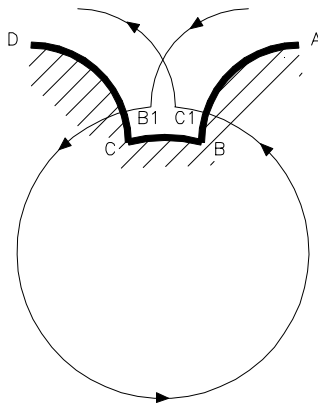
- b The contour from AB to BC is programmed. With active radius compensation the tool retracts along CD. If BC is smaller than two times the tool radius, the tool collides with the workpiece during the movement from B' to C' and from C' to D'.



- c A contour of the shape given in the illustration below is programmed. If the straight line is smaller than two times the tool radius, the tool collides with the workpiece during machining.



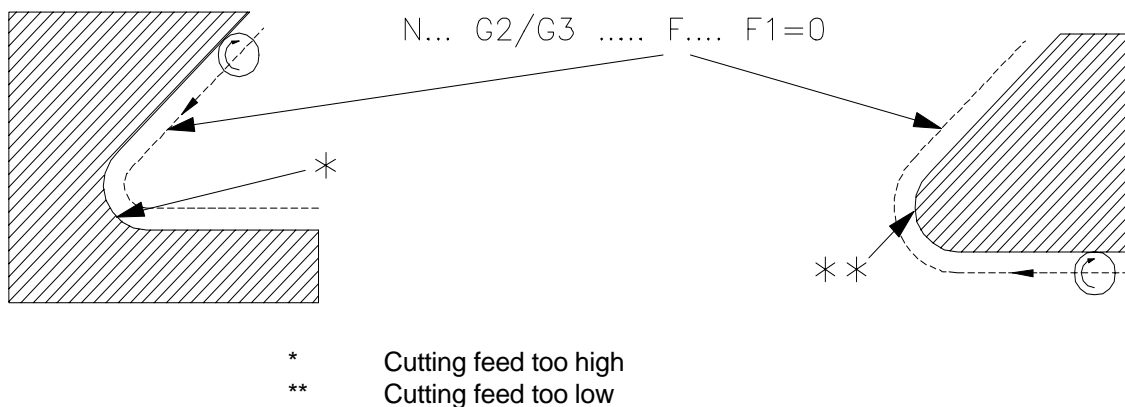
- d A contour of the shape given in the illustration below is programmed. The tool moves to point B1, then from B1 to C1 and then parallel along CD. The movement from B1 to C1 takes place in the same direction as programmed on the circle BC. If the circular movement BC is too small, this results in the tool making almost a complete circle before it arrives at C1.



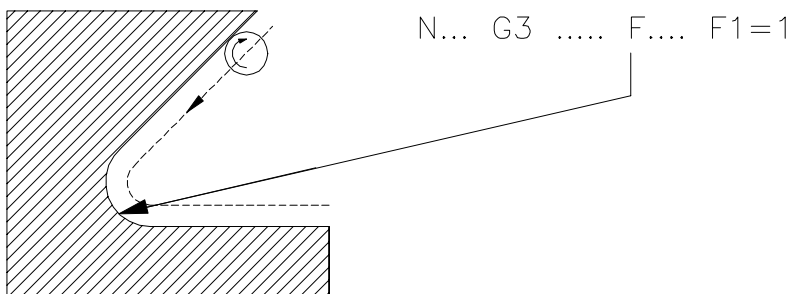
Constant cutting feed

The parameter 'F1=' is used to ensure that the programmed feedrate along a workpiece contour remains constant regardless of the radius of the mill and the contour shape. This controlled velocity is called the **CONSTANT CUTTING FEED**.

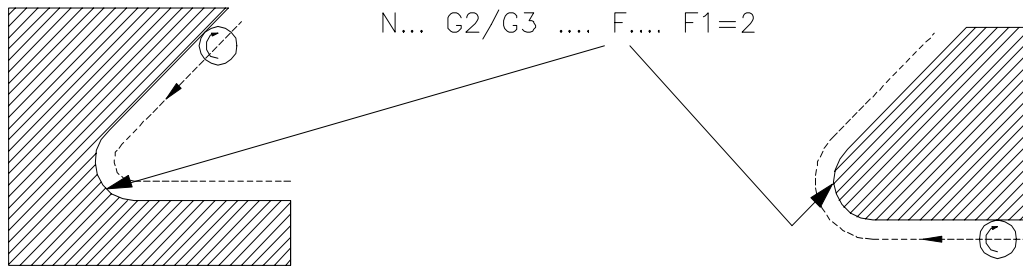
F1=0 Constant cutting feed not applied (default mode; also set at **CLEAR CONTROL** or **M30** or **Softkey CANCEL PROGRAM**). The programmed feedrate should be the velocity of the tooltip.



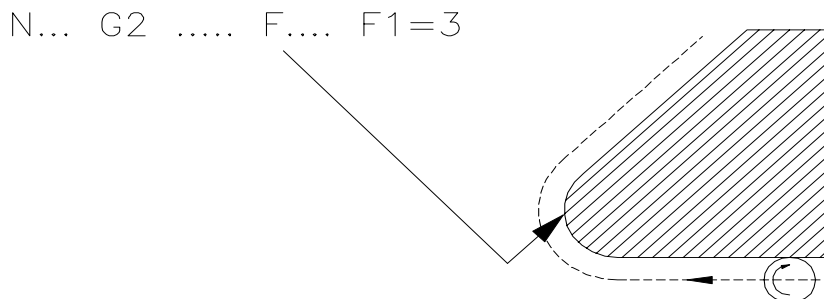
F1=1 Constant cutting feed applied only on the inside of arcs. The programmed feedrate is reduced to assure that the tooltip moves with the reduced velocity on the inside of an arc.



F1=2 Constant cutting feed applied on the inside and outside of arcs. The programmed feedrate is reduced (inside arc) or increased (outside arc) to assure that the tooltip moves with the recalculated velocity. If the increased velocity is greater than the maximum feedrate (a Machine Constant value) the maximum feedrate is used.



F1=3 Constant cutting feed applied only on the outside of arcs. The programmed feedrate is increased to assure that the tooltip moves with the increased velocity on the outside of an area. If the increased velocity is greater than the maximum feedrate (a Machine Constant value) the maximum feedrate is used.



Switching from one radius comp. function to another one

When switching from one function, e.g. G41 to G42, G43 or G44, the tool ends in a position, which is calculated with the first function active and starts in a position calculated with the other function active. When these two positions do not coincide a linear feed movement from one position to the other one is executed.

Ending radius compensation

The function G40 cancels the tool radius compensation. Thereafter programmed coordinates refer to movements of the tooltip.

Plane for radius compensation

The radius compensation is performed in the plane indicated by G17, G18 and G19.

Tool axis movement

A simultaneous movement of the tool axis and the axes of the main plane (defined with G17, G18 or G19) with activated radius compensation is possible. The compensated movements in the main axes can be linear or circular.

Helix interpolation

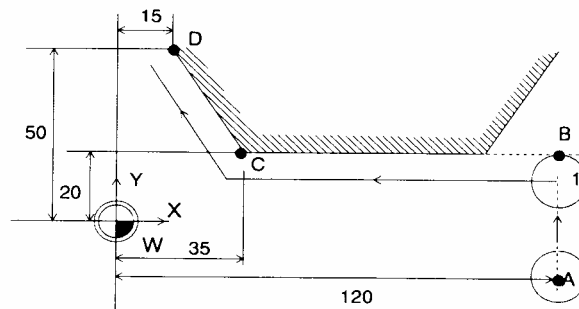
If helix interpolation is used, radius compensation can be used on the circular movement in the plane indicated with G17, G18 or G19.

Using the cylindrical coordinate system

When the cylindrical coordinate system (G182) is activated, the functions G41/G42/G43/G44 as described can also be used in the plane of the cylinder.

Examples

Example 1



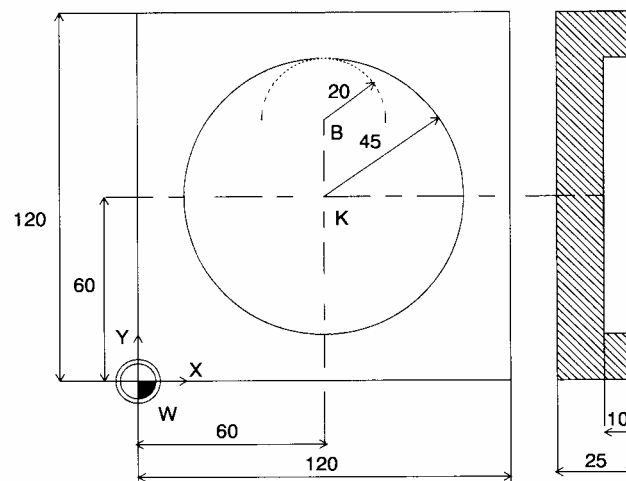
N21 G17
 N3 T1 M6
 N4 G0 X120 Y-20 Z5 S3000 M3
 N5 G1 Z-10 F600
 N6 G43
 N7 Y20
 N8 G41

Load tool 1 and its offsets. Mill diameter is 10 mm
 Start spindle, move the tool rapidly to point A
 Moving to depth with a feedrate of 600 mm/min.
 Set radius compensation to.
 Move the tool to point B.
 Select radius compensation RIGHT. Move the tool at set
 feedrate.
 Along the right hand side of the workpiece

N9 X35
 N10 X15 Y50
 N11 G40

Cancel radius compensation.

Example 2.



N2 G17
 N3 T1 M6
 N4 G0 X60 Y85 Z0 S3000 M3
 N5 G1 Z-10 F500
 N6 G43 X80 F300

N7 G41
 N8 G3 X60 Y105 R20
 N9 I60 J60
 N10 X40 Y85 R20
 N11 G40

Load the tool. The mill has a diameter of 10 mm.
 Start the spindle and move tool to starting point B.
 Feed the tool to depth.
 Move the tool to the starting point of the small circle. Set
 new feedrate to 300 mm/min.
 Set radius compensation LEFT.
 Move the tool with a circular movement, to enter the contour.
 Mill the complete circle.
 Exit the contour by using a small circular movement.
 Cancel the radius compensation.

5.24 G43/G44 Tool radius compensation to/past endpoint

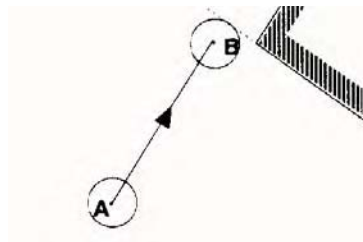
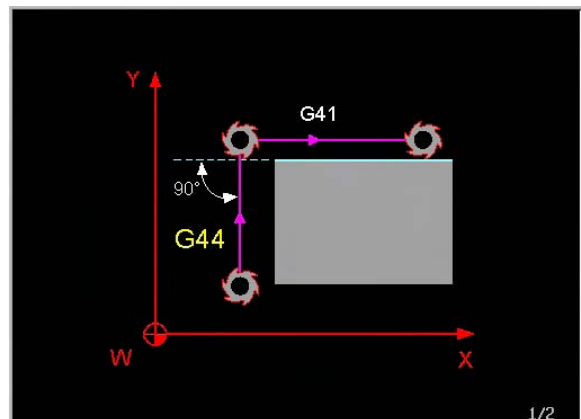
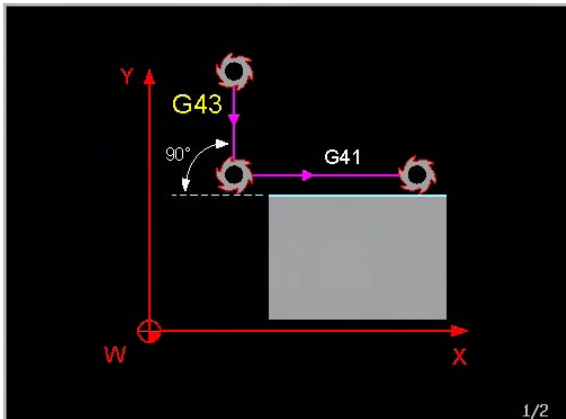
To move the tool with cutter radius compensation. TO/PAST a programmed position.

G43 activates radius compensation 'TO' a programmed position (tool radius is subtracted from the programmed position).

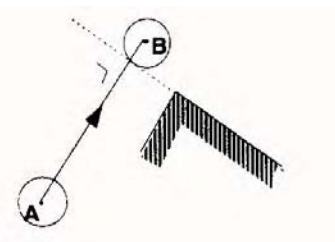
G44 activates radius compensation 'PAST' a programmed position (tool radius is added to the programmed position).

Format

G43/G44 {axis coordinates}



G43 'TO'



G44 'PAST'

Alternative

G43 is mostly used for axis parallel positioning movements. If the positioning movement is not parallel to the axis, starting point B should be calculated. It is therefore wiser to use the possibilities of G61 (tangential approach).

Notes and usage

Modality

This function is modal with G40, G41 and G42.

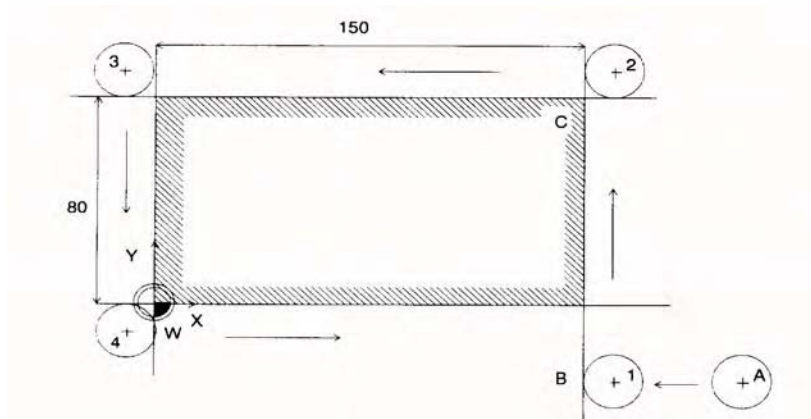
Circular movement

If a G43 or G44 is active with a circular movement (G2/G3), an error message is displayed. With circular movements G41 or G42 must be used.

Axis parallel movement

If a G43 or G44 is used with an axis parallel movement and only one coordinate is programmed, the position in that axis is calculated. The other axis remains unchanged. So G43/G44 only can be used with radius compensation and axis parallel movements.

Example 2



```

N9 T1 M6
N10 G0 X200 Y-20 Z-5 S1000 M3

N11 G43 X150
N12 G1 F200
N13 G44 Y80

N14 X0
N15 Y0
N16 X150

N17 G40 Y-20
N18 G0 X200

```

Load tool 1 and its offsets.
 Make spindle rotate clockwise at 1000 rev/min, move the tool to position A and then at depth.
 Move the tool rapidly to point B.
 Set linear feedrate to 200 mm/min.
 Move the tool along the Y-axis PAST edge Y80 (point 2). The function G44 remains active in the blocks that follow.
 Move the tool along the X-axis PAST edge X0 (point 3).
 Move the tool along the Y-axis PAST edge Y0 (point 4).
 Move the tool along the X-axis PAST edge X150. The tool is free from the part
 Cancel the radius compensation.
 Rapid traverse movement to position A.

5.25 G45 Axis parallel measuring movement and measuring tool dimensions

Note

Use of this function is limited only to programs made on earlier control systems.

The G45 function operates only parallel to the axis. G145 has improved functionality and is also able to perform measurements, which are not parallel to the axis. It is therefore wiser to use the new G145 basic measurement movement.

Two actions can be done with G45:

- 1) Measuring a point with only G45
- 2) Measuring tool dimension G45 + M25

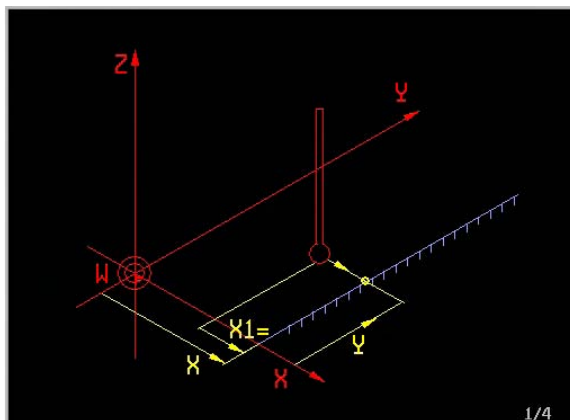
5.25.1 G45 Measuring a point

To measure the actual axis coordinate with a touch trigger probe when being moved in the axis to the programmed position. This allows the difference between the actual and programmed position to be used to check the dimensional accuracy of the workpiece.

Format

G45 [Measuring position] {I+/-1} {J+/-1} {K+/-1} {L+/-1} {X1=...} {N=...} {E...}

The plane for the rotary table is determined by the definition of the 4th axis in the machine constant list. (MC117 should be 4 and MC118 should be B (66) or C (67)). L relates to the 4th axis B or C. Rotary axis A is not allowed.



G Measuring a point
 X Measurement target coordinate
 Y Measurement target coordinate
 Z Measurement target coordinate
 B Measurement point angle
 C Measurement target angle
 I Measurement direction for X axis
 J Measurement direction for Y axis
 K Measurement direction for Z axis
 L Measurement direction rotary-axis
 M M25 for tool measurement
 E Parameter-nr measured coordinate
 N= Point-nr.for measured coordinate
 X1= Measurement path length
 ?90= Measurement target abs. (X,Y,Z..)

?91= Measurement target incr.(X,Y,Z..)
 P1= Point definition number

Measuring position

X, Y, Z Measurement target coordinate
 C Measurement target angle
 P Point definition number

Measuring parameters

I Measurement direction for X axis
 J Measurement direction for Y-axis
 K Measurement direction for Z-axis
 L Measurement direction rotary-axis
 X1= Measurement path length

Measuring results

E Parameter number measured coordinate
 N Point number for measured coordinate

The difference between the measured and programmed coordinate is calculated and stored internally for use with G49 or G50.

Notes and usage

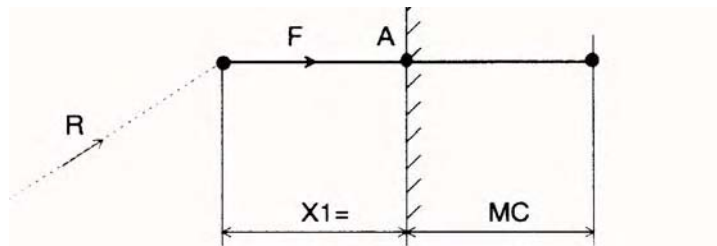
Associated functions

G46, G49, G50 and the basic measuring movement G145
 M25, M27, M28

Measuring position

The programmed coordinates specify the point to be measured.

Pre- and post-measuring distance (X1=)



The pre-measuring distance defines the position in the axis to be measured from where the movement with the measuring feed starts. This distance is programmed with the word X1=.

If X1= is not programmed, a Machine Constant value (MC844) is used.

With the post-measuring distance (MC845) is defined how far the probe can pass the programmed position of the axis before it is triggered.

Measuring sequence

1. The probe moves rapidly to the pre-measuring position, which is defined, by the programmed position and the pre-measuring distance in the axis to be measured. This movement is executed with the positioning logic of G0.
2. After the probe reaches the pre-measuring point it moves at a fixed feedrate (MC843) along the indicated axis in the programmed direction towards the programmed position in the axis. The probe can pass this point, but must be triggered along the path between the pre- and post-measuring distance.
3. When the probe touches the workpiece, the measured coordinate is stored and the probe rapidly moves back to the pre-measuring position.

Storing measuring result (E, N=)

The measured coordinate can be stored in either the E-Parameter Memory (E) or and the Point Memory (N=).

Storing the coordinate in an E-parameter has the advantage of allowing additional calculations to be performed, such as in a macro.

The difference between the measured and programmed coordinate is calculated and stored internally for use with G49 or G50.

The stored differences are cancelled as soon as a new measuring function (G45 or G46) is activated or with Softkey CLEAR CONTROL or CANCEL PROGRAM.

Error messages

An error message is displayed and the movement stops,

1. If the probe touches an obstruction during the rapid movement to the pre-measuring position,
2. If the probe exceeds the post-measuring distance.

Collision protection

As soon as the measuring probe is triggered during any other movement than the actual feed movement for the measurement it, an error is generated and the movement interrupted. Sometimes the probe is triggered due to very fast movements and not by a real collision.

A machine constant (MC850) can be used to store the information that collision protection

- Is switched off during the measurement movement and possibly during retraction after measurement;
- Is effective during all movements or only during feed movements.

Tool memory

The radius of the probe and its length are stored in the tool memory together with a tool number. Tool type Q3=9999 can be entered to indicate the measuring probe.

Example: P5 T5 Q3=9999 L150 R4

When tool T5 is called with Q3=9999 the control system recognizes this tool as the measuring probe. The probe radius is called and used to correct the measurement position.

If a function for spindle direction (M3 or M4) is entered, this function is suppressed and an error message displayed.

Air blow before measuring

To clean the workpiece at the position to be measured an air blow can be executed during a fixed time (MC842).

The air blow is activated once by an M-function and executed each time a pre-measuring position is reached. Refer to the machine tool builder's documentation for the number of the M-function for the air blow.

Restriction

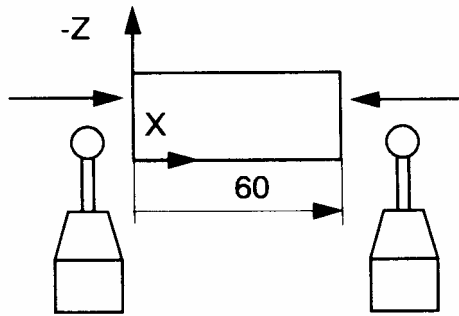
1. Only one axis coordinate can be measured in a G45-block.
2. An omni directional probe should be used.
3. In the tool axis the probe is only triggered if pressed. This means that a measurement in the positive direction of the tool axis is not possible.

Note: The G45 function is also used in conjunction with the M25 function for measuring tool dimensions.

Refer to G45 + M25 section for additional information.

Example

Example 1 Measuring a point in X-axis



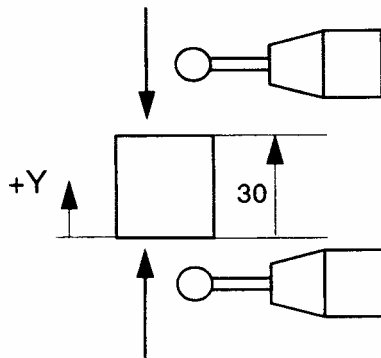
Measuring in positive direction

N.. G45 X0 Y20 Z-10 I1 E1 N=1

Measuring in negative direction

N.. G45 X60 Y20 Z-10 1-1 E1 N=1

Example 2 Measuring a point in Y-axis



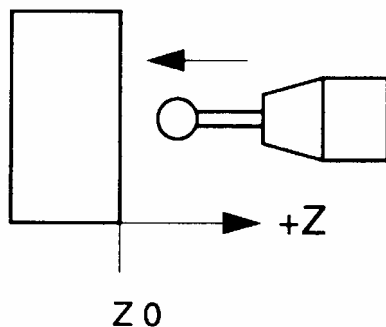
Measuring in positive direction

N.. G45 X30 Y0 Z-10 J1 E1 N=1

Measuring in negative direction

N.. G45 X30 Y30 Z-10 J-1 E1 N=1

Example 3 Measuring a point in Z-axis



Measuring in negative direction

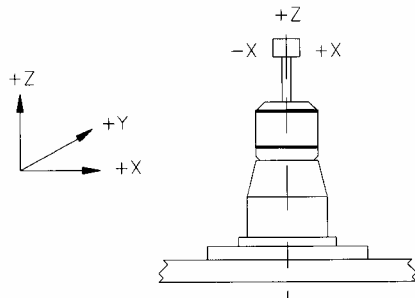
N.. G45 X30 Y30 Z0 K-1 E1 N=1

The point is measured; the measured position calculated and stored in Point Memory location 1 and parameter E1.

Note: Measuring in the tool axis is only possible in the negative direction.

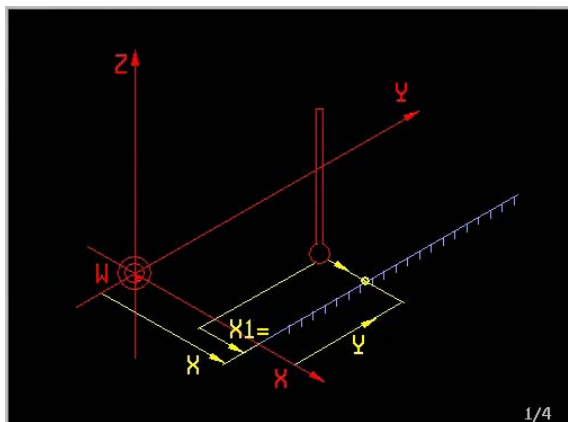
5.25.2 G45 + M25 Measure tool dimensions

To measure tool dimensions using an square-head measuring probe.



Format

G45 {I...} {J...} {K...} {X1=...} M25



G Measuring a point
 X Measurement target coordinate
 Y Measurement target coordinate
 Z Measurement target coordinate
 B Measurement point angle
 C Measurement target angle
 I Measurement direction for X axis
 J Measurement direction for Y axis
 K Measurement direction for Z axis
 L Measurement direction rotary-axis
 M M25 for tool measurement
 E Parameter-nr measured coordinate
 N= Point-nr.for measured coordinate
 X1= Measurement path length
 ?90= Measurement target abs. (X,Y,Z..)

?91= Measurement target incr.(X,Y,Z..)
 P1= Point definition number

Measuring parameters

I Measurement direction for X axis
 J Measurement direction for Y-axis
 K Measurement direction for Z-axis
 L Measurement direction rotary-axis
 X1= Measurement path length

Notes and Usage

Associated functions

G45, G46, G49, G50
 M26, M27, M28

Measuring tool dimensions

An square-head-measuring probe mounted at a fixed position on the machine tool, is used for measuring the tool dimensions.

Measuring in the tool axis gives the tool length.

Measuring in two directions of the same axis gives the tool radius.

Position of the square-head probe

The position of the square-head measuring probe (MC3155, MC3?55) and its width are red in the Machine Constant Memory (MC847).

Measuring sequence

The measurements are executed in the same way as with G45. Instead of programming the position of the fixed probe, its coordinates are picked up from the Machine Constant Memory.

Updating the tool memory

The tool memory is updated with the function G50. Refer to that function for details.

Note

1. Tool measurement can also be performed on the control in the mode OPERATE. Refer to the Operating Manual for details.
2. Refer to G145 for an example for automatic measurement of the tool dimensions with a measuring box.

Example Measuring tool length

N89 T1 M6

N90 G45 K-1 X1=5 M25

N91 G50 T1 L1=1

Load tool 1

Measure the tool length in the negative direction of the Z-axis. Pre-measurement distance is 5 mm.

Correct the tool length of tool 1 in the tool memory

5.26 G46 Measuring a full circle or probe calibration

Note

Use of this function is limited only to programs made on earlier control systems.

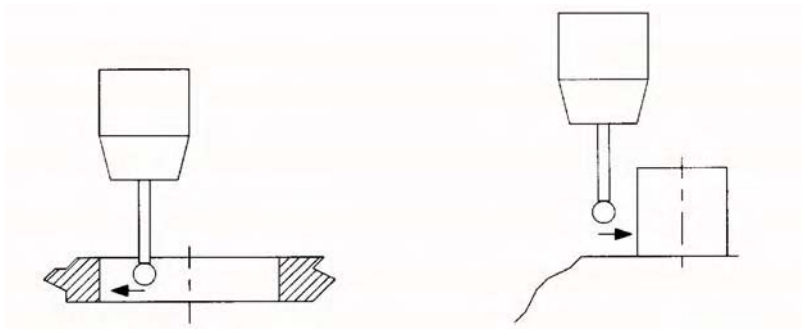
The G46 function operates only parallel to the axis. G146 has improved functionality and is also able to perform measurements, which are not parallel to the axis. It is therefore wiser to use the new G145 basic measurement movement.

Two actions can be done with G46:

- G46 Measuring a full circle with only G46
- Probe calibration with G46 + M26

5.26.1 G46 Measuring a full circle

To measure a full circle and to determine the centre point coordinates and any deviation between the programmed circle radius and the calculated radius.



Measuring an inner circle

measuring an outer circle

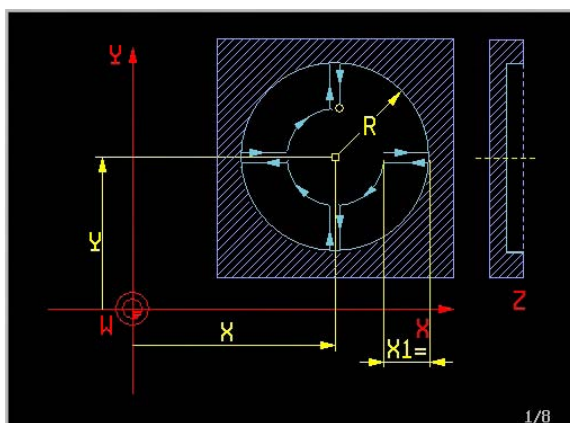
Format

Measuring an inner circle

G46 [Circle centre coordinates.] R... {1+1 J+1} {1+1 K+1} {J+1 K+1} {F...} {X1=...} N=... E...

Measuring an outer circle

G46 [Circle centre coordinates.] R... {1-1 J-1} {1-1 K-1} {J-1 K-1} {F...} {X1=...} N=... E...



G Measuring a circle
 X Center point coordinate
 Y Center point coordinate
 Z Center point coordinate
 B Measurement target angle
 C Measurement target angle
 I Measurement direction for X axis
 J Measurement direction for Y axis
 R Circle radius
 M M26: probe radius measurement
 E Parameter-nr. measured radius
 N= Point-nr.measured centre point
 X1= Measurement path length
 ?90= Centre point abs. (X,Y,Z...)
 ?91= Centre point incr. (X,Y,Z...)

P1= Point definition number

Circle parameters

X, Y, Z Centre point coordinate
 C Measurement target angle
 P Point definition number
 R Circle radius

Measuring parameters

I Measurement direction for X axis
 J Measurement direction for Y-axis
 K Measurement direction for Z-axis
 F Feed between measurements
 X1= Measurement path length

Measuring results

E Parameter number measured radius
 N= Point number measured centre point

Notes and Usage

Associated functions

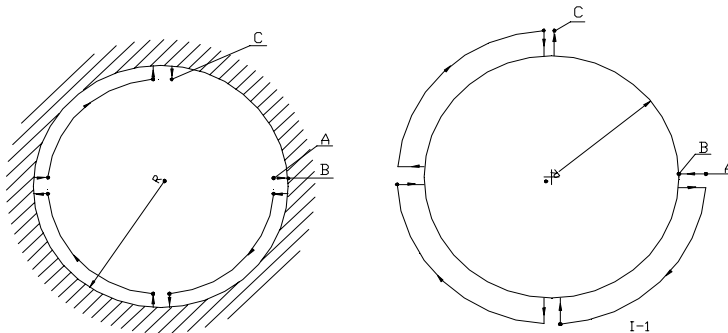
G45, G49, G50
 M26, M27, M28

The G46 function is for measuring a full circle. Under the linear measurement movement G145 a macro is describes, with which a full circle can be measured.

Measuring the positions

Four positions are measured when a G46 block is executed. The measurements take place as if four G45 blocks were programmed. So refer to G45 for additional information about PRE- AND POST-MEASURING DISTANCE, TOOL MEMORY, AIR BLOW and COLLISION PROTECTION.

Measuring sequence



A= Start point (is positioned with rapid speed)
 B= Measurement point
 C= End point

1. The probe moves rapidly to the pre-measuring position of the first point to be measured. This position is defined by the programmed circle centre, the programmed radius and the pre-measuring distance (X1=). This movement is executed with the positioning logic of G0.
2. The probe moves at a fixed feedrate (MC 843) towards the first point on the programmed circle. The probe can pass the point, however, it must be triggered along the path between the pre- (MC 844) and post-measuring (MC 845) distance.
3. When the probe is correctly triggered, the measured position is automatically stored. Then the probe moves back rapidly to the starting position and with the programmed feedrate (F-word) along the circle in a clockwise direction until it reaches the second pre-measuring position.
4. The procedure just given is repeated for the second, third and fourth position.

5. When the fourth position has been measured, the circle centre and radius are calculated from the four measured points. The coordinates of the circle centre are stored in the Point Memory and the radius in the E- parameter memory.

Measuring inner or outer circle (I/J/K)

Any pair of the addresses I, J, K simultaneously define the type of circle to be measured and the plane in which the circle is located. A pair of addresses must be stated in each G46-block.

Plane	Inner Circle		Outer Circle	
XY (G17)	I+1	J+1	I-1	J-1
XZ (G18)	I+1	K+1	I-1	K-1
XZ (G19)	J+1	K+1	J-1	K-1

Storing centre point coordinates (N=)

The word N= states the number in the Point Memory where the calculated coordinates of the centre point are stored. E.g. N=12, means that the centre point coordinates are stored in P12.

Storing the circle radius (E)

The E-word states the number of the E-parameter where the calculated radius is stored. E.g. E45 means that the circle radius is stored as the value of E-parameter 45.

Error messages

An error message is displayed and the movement stops,

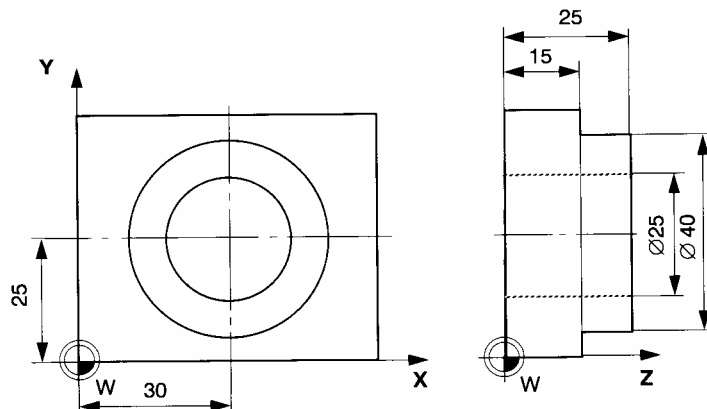
1. If the probe touches an obstruction during the movement to the premeasuring position,
2. If the probe exceeds the post-measuring distance.

Restriction

Each G46-block can measure only one circle.

Note: The G46 function is also used in conjunction with the M26 function for probe calibration. Refer to G46 + M26 section for additional information.

Example Measuring an inner and outer circle in the XY-plane



Measuring the inner circle:

N... G46 X30 Y25 Z20 I+1 J+1 R12.5 F3000 N=59 E24

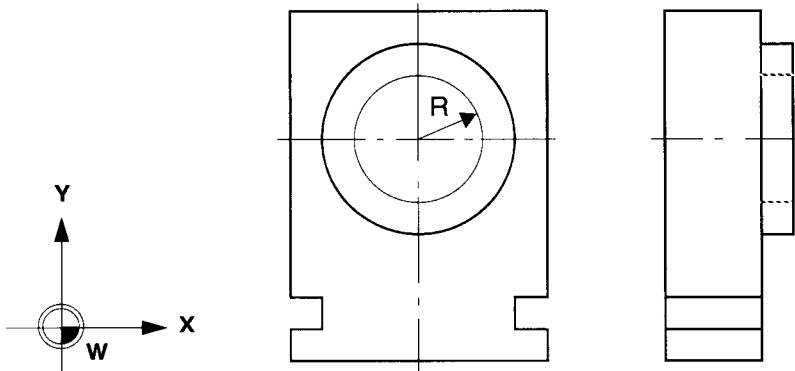
Measuring the outer circle:

N... G46 X30 Y25 Z20 I-1 J-1 R20 F3000 N=58 E23

The circles are measured, the centre points of the measured circles calculated and stored in the point memory and the radius calculated and stored in the parameter memory.

5.26.2 G46 + M26 Probe calibration

To determine the radius of a touch trigger probe by touching a calibration ring, thus a ring gauge whose diameter is exactly known.



A probe must be calibrated:

- when the probe is used for the first time
- when a new stylus is used
- after any suspected bending of the stylus.

Note: It is assumed that the position of the ball centre relative to the spindle axis is already determined.

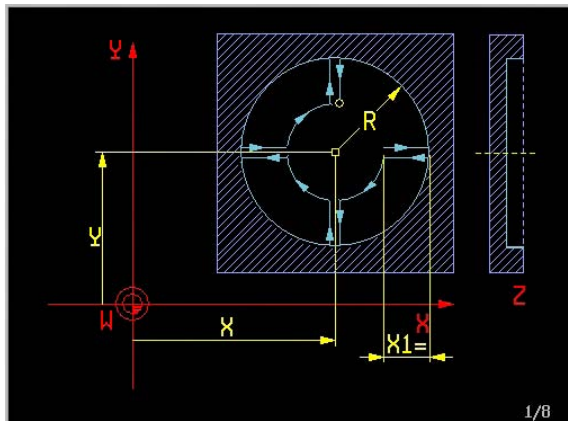
Format

Measuring an inner ring gauge

G46 {1+1 J+1} {I+1 K+1} {J+1 K+1} {F...} {X1=...} M26

Measuring an outer ring gauge

G46 {I-1 J-1} {I-1 K-1} {J-1 K-1} {F...} {X1=...} M26



G Measuring a circle
 X Center point coordinate
 Y Center point coordinate
 Z Center point coordinate
 B Measurement target angle
 C Measurement target angle
 I Measurement direction for X axis
 J Measurement direction for Y axis
 R Circle radius
 M M26: probe radius measurement
 E Parameter-nr. measured radius
 N= Point-nr.measured centre point
 X1= Measurement path length
 ?90= Centre point abs. (X,Y,Z..)
 ?91= Centre point incr. (X,Y,Z..)

P1= Point definition number

Measuring parameters

I Measurement direction for X axis
 J Measurement direction for Y-axis
 K Measurement direction for Z-axis
 F Feed between measurements
 X1= Measurement path length

Notes and Usage

Associated functions

G45, G46, G49, G50
M25, M27, M28

Position of the calibration ring

The position of the calibration ring and its radius are stored in the Machine Constant Memory.

Measuring sequence

The measurements are executed in the same way as with G46. Instead of programming the position of the ring gauge, its coordinates are picked up from the Machine Constant Memory.

Measuring inner or outer ring gauge (I/J/K)

Any pair of the addresses I, J, K simultaneously define the type of ring to be measured and the plane in which the ring is located. A pair of addresses must be stated in each G46-block.

Plane	Inner ring		Outer ring	
XY (G17)	I+1	J+1	I-1	J-1
XZ (G18)	I+1	K+1	I-1	K-1
XZ (G19)	J+1	K+1	J-1	K-1

Updating the radius of the probe

The difference between the radius of the ring stored in the Machine Constants and the measured radius is used to update the probe radius and store this value in the tool memory for the active tool (= the probe).

When probe calibration is required

A probe should be calibrated in the cases mentioned above and also:

- If accuracy demands it.
- If the repeatability of relocation of the probe in the spindle is poor. In this case calibrating may be required each time the probe is selected.

Example

N46002
N1 G17
N2 T1 M6
N3 D207 M19
N4 G46 I1 J1 M26 F3000

Set the plane of operation to be the XY-plane
Load the touch trigger probe
Stop the spindle in a defined position
Calibrate the probe by moving it to the inside surface of a ring gauge located in the XY- plane. The measured radius of the touch probe is stored in the tool memory location of the active tool (T1). A default MC value is used for the pre-measuring distance.

N5 Z200 M30

5.27 G49 Checking on tolerances

Note

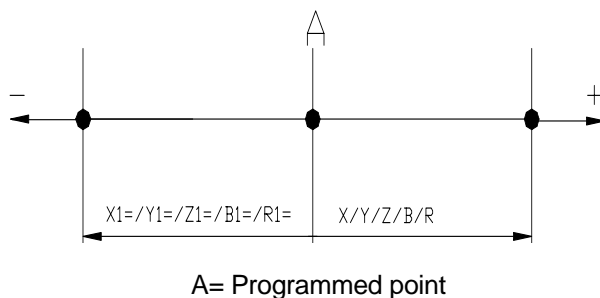
Use of this function is limited only to programs made on earlier control systems.

Since the G45 uses an internal memory, the G49 function can only be used together with G45. Basic measurement movement G145 uses E-parameters and may have the same functionality. It is therefore wiser to use the new basic measurement movement G145.

Purpose

To check whether the difference between a programmed value and the measured (G45/G46) value lies within set tolerance limits. If the difference is within the limits the program is allowed to continue. However, if the difference is not within limits there can be:

- a repeat of a section of the program until the difference is acceptable
- a conditional jump in the program
- a display of an error message.



The measured point must lie within the highest tolerance limit (X/Y/Z/B/R) and the lowest tolerance limit (X1 =/Y1 =/Z1 =/B1 =/R1 =).

Format

A repeat of a program section

G49 {X..., X1=...} {Y..., Y1=...} {Z..., Z1=...} {B..., B1=...} {R..., R1=...} N1=... {N2=...} {E..}

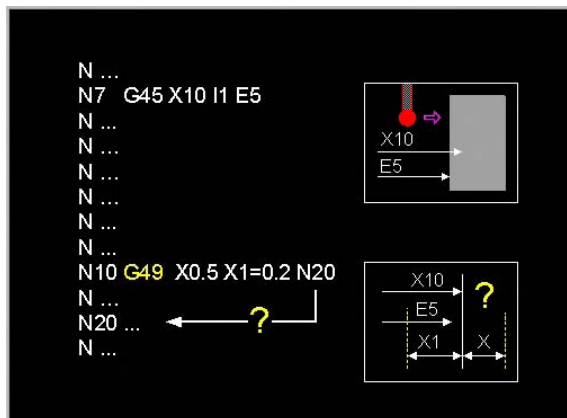
The plane for the rotary table is determined by the definition of the 4th axis in the machine constant list. (MC117 should be 4 and MC118 should be B(66) or C(67)). B, B1 relates to the 4th axis B or C. R applies to the plane of the rotary table. Rotary axis A is not allowed.

A conditional jump.

G49 {X..., X1=...} {Y..., Y1=...} {Z..., Z1=...} {B..., B1=...} {R..., R1=...} N=... E...

Display an error message

G49 {X..., X1=...} {Y..., Y1=...} {Z..., Z1=...} {B..., B1=...} {R..., R1=...}



G Checking on tolerances
 X Positive tolerance value in X
 Y Positive tolerance value in Y
 Z Positive tolerance value in Z
 B Positive tolerance value in B
 C Positive tolerance value in C
 R Positive tolerance circle radius
 E Jump condition: $E > 0$
 N= Jump to blocknumber
 N1= Repeater begin block
 N2= Repeater end block
 X1= Negative tolerance value in X
 Y1= Negative tolerance value in Y
 Z1= Negative tolerance value in Z
 B1= Negative tolerance value in B

C1= Negative tolerance value in C
 R1= Negative tolerance circle radius

Tolerance values

X Positive tolerance value in X
 X1= Negative tolerance value in X
 Y Positive tolerance value in Y
 Y1= Negative tolerance value in Y
 Z Positive tolerance value in Z
 Z1= Negative tolerance value in Z
 B Positive tolerance value in B
 B1= Negative tolerance value in B
 R Positive tolerance circle radius
 R1= Negative tolerance circle radius

Conditional jump

E Jump condition: $E > 0$
 N= Jump to block number

Repeat of program section

N1= Repeater begin block
 N2= Repeater end block

Notes and Usage

Associated functions

G45, G46, G50

Note If the difference between the measured value and the programmed value is within the set tolerances, the program continues with the block after the G49 or G50.

Continuation if values are within limits

If the difference between the measured value and the programmed value is within the set tolerances, the program continues with the block after the G49.

Repeat of a program section (E, N1=, N2=)

The words E, N1= and N2= are used to repeat a section of the program when a tolerance limit is exceeded.

The E-word specifies the number of repeats ($E > 0$).

When no number of repeats is programmed (no E-word is present), the sequence is repeated only once.

Block numbers of repeat sequence (N1=, N2=) '

These block numbers must be in the same part program or subprogram.

If N2= is not programmed, only the block indicated by N1= is repeated the specified number of times.

Order of blocks to be repeated

The order of executing the blocks in the repeat sequence must be the same as the order originally programmed. So in the program block N1=.. must be before block N2=..

Continuation after the repeat

Once the repeats are executed, the program continues with the block after the G49.

Conditional jump (N=, E)

The words N= and E are used to specify a conditional jump when a tolerance value is exceeded.

The value of parameter E must be greater than zero before a jump can occur.

The word N= states the block number in the same program or subprogram to which control will jump when E>0.

Continuation if no jump is executed

If no jump is executed, because the limits are not exceeded or E≤0, program execution continues with the block after G49.

Error message

An error message is generated by the CNC, if the measured value exceeds a tolerance limit and neither a repeat of a program section nor a conditional jump is programmed.

Continuation after an error message

After resetting the error program execution continues with the block after G49.

Checking on the highest and lowest tolerance limit

If tolerance checks are used to see if a part is made within tolerances, two G49-blocks can be used. The order must be:

1. Check to see if the highest tolerance limit is exceeded. If this occurs the part is too big, so a jump out of the measuring section of the program is necessary.
2. Check to see if the lowest tolerance limit is exceeded. If this occurs the part is too small and has to be milled again, so a repeat of the milling section with an updated tool radius is necessary.

Examples

Example 1 A repeat of a program section

N97 G49 X0.005 X1=0.002 N1=80 N2=95 E2 If the measured position is more than 0.005 mm higher or 0.002 mm lower than the programmed position, the program section from block number N80 to N95 is repeated two times. After the repeat program execution continues from the block after N97.

Alternative:

E2 is the desired position

E3 is measured position due to G145.

N97 G29 E0 E0=E3<(E2-0.002) N=100 Jump to N100 when measure position smaller is then 0.002 mm.

N98 G29 E0 E0=E3>(E2+0.005) N=100 Jump to N100 when measure position greater is then 0.005 mm.

N99 G14 N1=80 N2=95 J2 Repeat program twice.

Example 2 A conditional jump

N197 G49 X0.005 X1 =0.002 E10 N=80 If the measured position is more than 0.005 mm higher or 0.002 mm lower than the programmed position and the value of parameter E10 is greater than zero, a jump in the program to block N80 is performed and program execution continues from that block.

Alternative:

E2 is the desired position.

E3 is the measured position due to G145.

N96 G29 E10 N=99

N97 G29 E0 E0=E3<(E2-0.002) N=80

N98 G29 E0 E0=E3>(E2+0.005) N=80

N99 ...

Example 3 A conditional jump and repeat of program section

N10 G49 R.02 R1=2 E1 N=13 E1=1 If the measured position is more than 0.02 mm higher than the programmed position, jump to N13. R1= is set high to avoid that this limit is exceeded.

N11 G49 R2 R1=.02 N1=1 N2=6 If the measured position is more than 0.02 mm lower than the programmed position, repeat the program section from N1 to N6. R is set high to avoid that this limit is exceeded.

Alternative:

E2 is the desired position of the circle radius.

E3 is measured position of the circle radius

N10 G29 E0 E0=E3>(E2+0.02) N=80

N98 G29 E0 E0=E3>(E2-0.02) N=95

N99 G14 N1=1 N2=16

5.28 G50 Processing measuring results

Note

Use of this function is limited only to programs made on earlier control systems.

Since the G45 uses an internal memory, the G50 function can only be used together with G45. Basic measurement movement G145 uses E-parameters and may have the same functionality. It is therefore wiser to use the new basic measurement movement G145.

Purpose

To make corrections derived from the measured differences on either the zero offsets or the tool dimensions.

Format

To change zero offsets

With standard zero offsets or MC84=0:

G50 {X1} {I...} {Y1} {J...} {Z1} {K...} [{B1}{C1}{C2}] [{B1=...}{C1=...}] {L...} N=...

With MC84>0 zero offsets extends:

G50 {X1} {I...} {Y1} {J...} {Z1} {K...} [{B1}{C1}{C2}] [{B1=...}{C1=...}] {L...} **N=54.[nr]**

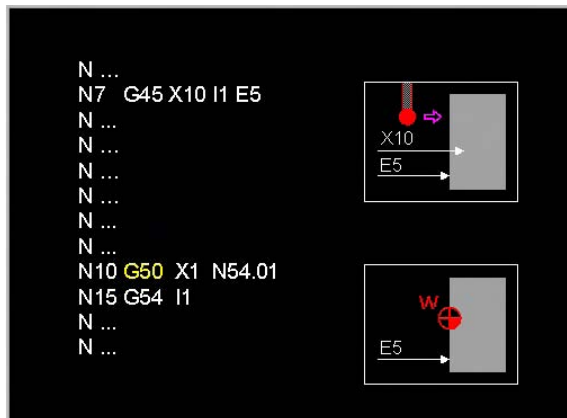
The plane for the rotary table is determined by the definition of the 4th axis in the machine constant list. (MC117 or MC120 should be 4 and the associated axes should be B(66) or C(67)). B1 or C1 relates to the 4th axis B or C. Rotary axis A is not allowed.

To change the tool length

G50 T... L1=1 {I...} {J...} {K...} {T2=...}

To change the tool radius.

G50 T... R1=1 {X1=...} {T2}



G Processing measuring results
 X 1=zero point shift in X
 Y 1=zero point shift in Y
 Z 1=zero point shift in Z
 B 1=zero point shift in B
 C 1=zero point shift in C
 I Multiplication factor for X
 J Multiplication factor for Y
 K Multiplication factor for Z
 L Multipl. factor for rotary-axis
 T Tool dimensions to be corrected
 N= Offset-nr for correction (52-59)
 X1= Multiply factor for tool radius
 B1= Prog.angle in B after calculation
 C1= Prog.angle in C after calculation

L1= 1=correction of tool length
 R1= 1= correction of tool radius

Zero offsets

N= Offset-nr for correction (52-59)
 X X1: zero point shift in X
 Y Y1: zero point shift in Y
 Z Z1: zero point shift in Z
 B B1: zero point shift in B
 C C1: zero point shift in C
 I Multiplication factor for X

J	Multiplication factor for Y
K	Multiplication factor for Z
L	Multiply factor for rotary-axis
B1=	Prog. angle in B after calculation
C1=	Prog. angle in C after calculation
Tool dimensions	
T	Tool dimensions to be corrected
X1=	Multiply factor for tool radius
L1=	L1=1: correction of tool length
R1=	R1=1: correction of tool radius

Notes and Usage

Associated functions

G45, G46, G49

To change offset values (N=)

With the G50 function new offset values derived from the measured corrections can be stored in the Zero Offset Memory.

Multiplication factor for axes (I, J, K,)

A multiplication factor can be applied to the measured difference, e.g. K8, means multiply Z-axis difference by 0.8.

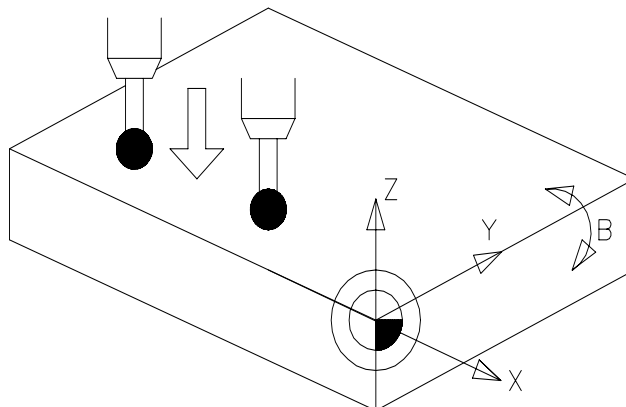
The multiplication factor can have a positive or negative value.

If no factor is stated, the default value +1 will be used automatically by the CNC.

Machine configurations (B1, C1, C2)

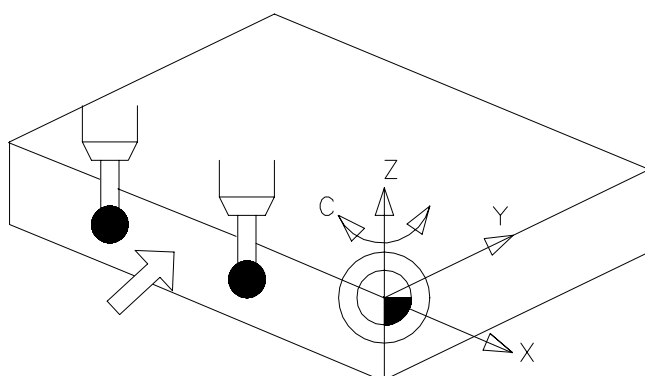
B-Axis B1: Aligning of a workpiece, mounted on a round table (B-axis) turning around the Y-axis, the measurement of two point in X direction are sufficient.

- The rotations angle is in respect with the X-axis.
- The workpiece is turning around the Y-axis.
- The measuring tracer stands in the Z-or Y-direction.



C-Axis C1: Aligning of a workpiece, mounted on a round table (C-axis) turning around the Z-axis, the measurement of two point in X direction are sufficient.

- The rotations angle is in respect with the X-axis.
- The workpiece is turning around the Z-axis.
- The measuring tracer stands in the Z-direction.



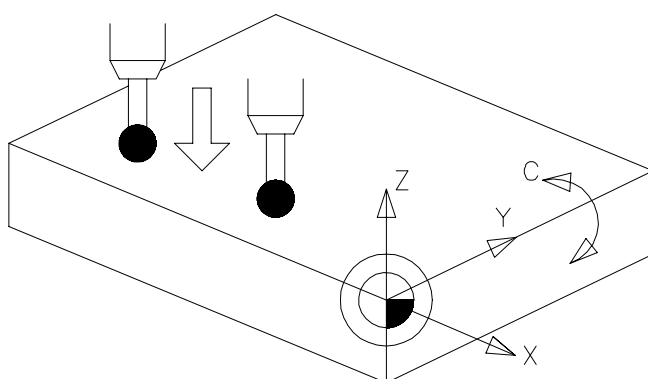
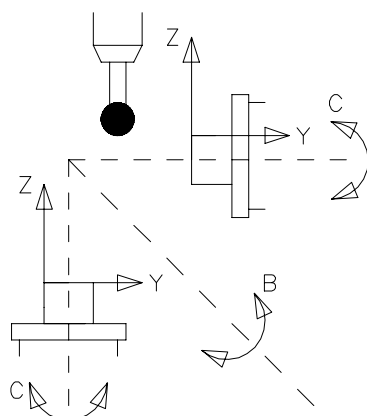
C-Axis C2:

1

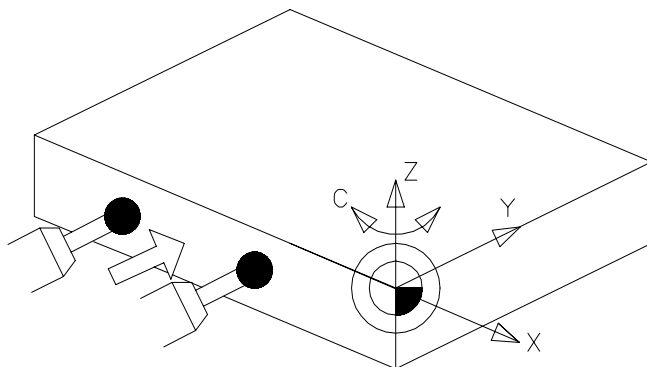
This is an extended possibility of C1:

The C-axis is turned 90 degrees and rotates around the Y-axis, instead of round the Z-axis. Alignment of a workpiece, mounted on a round table (C-axis) turning around the Y-axis, the measurement of two point in X direction are sufficient.

- The rotations angle is in respect with the X-axis.
- The workpiece is turning around the X-axis.
- The measuring tracer stands in the Z-direction.



- 2 Alignment of a workpiece, mounted on a round table (C-axis) turning around the Z-axis, the measurement of two point in X direction are sufficient.
- The rotations angle is in respect with the X-axis.
 - The workpiece is turning around the X-axis.
 - The measuring tracer stands in the Y-direction.



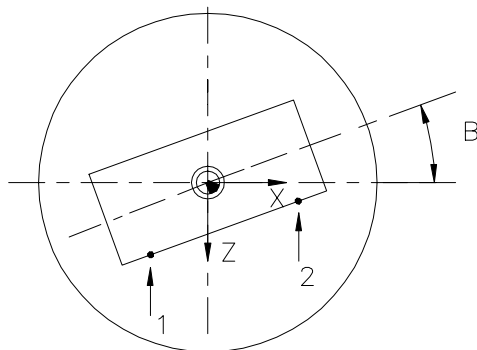
Aligning a workpiece on a rotary table (B1=, C1=)

If a workpiece is mounted on a table, which rotates around the Y- or Z-axis, it is possible to align the workpiece by measuring two points in the X- and Z- or X- and Z- direction.

The angle the workpiece makes with the X-axis or Y-axis is automatically calculated by the control and can be used to rotate the table, so that the workpiece is parallel to the X-axis or Y-axis.

If the workpiece makes initially an angle with the X-axis, this angle can be programmed with the word B1= or C1=.

If B1= or C1= is not programmed, B1=0 or C1=0 is assumed.



To change tool dimensions (T)

With the G50 function new tool dimensions derived from the measured corrections can be stored in the Tool Memory.

Multiplication factor for tool dimensions (I, J, K, X1=)

The multiplication factor for the tool radius is X1=.

The multiplication factor for the length correction depends on the active main plane defined by G17, G18 or G19:

K...	for Z-difference	(G17-plane is active)
J...	for Y-difference	(G18-plane is active)
I.	for X-difference	(G19-plane is active)

The multiplication factor can have a positive or negative value.

If no factor is stated, the default value +1 will be used automatically by the CNC.

Examples**Example 1** Changing a stored zero offset

N... G50 X1 I0.8 N=54

Change the X-coordinate of the G54 offset by multiplying the correction by 0.8 and storing the new G54 X-coordinate value into the offset memory.

Example 2 Changing a tool dimension

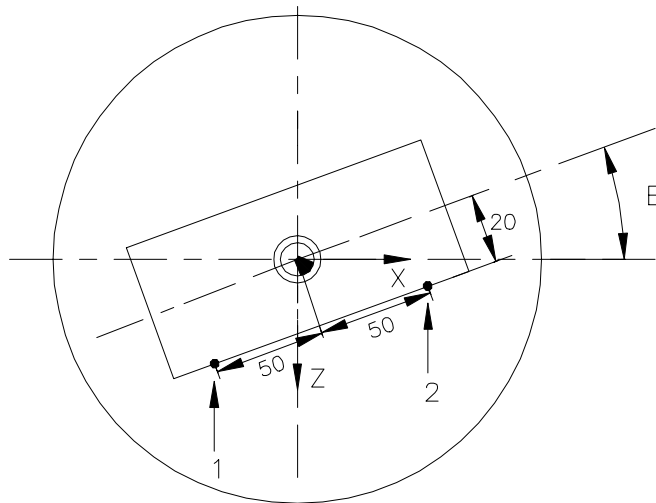
N... G50 T5 L1=1 K0.97 R1=1

Change the length of tool 5 by multiplying the Z-difference (tool in Z-axis) by 0.97, and store the new dimension into the tool memory.

Example 3 Aligning a workpiece mounted on a rotary table

A part is mounted on a rotary table and should be aligned parallel to the X-axis. With a touch trigger probe two points on the part are measured and then the table is rotated over the calculated angle.

The Controller knows, when G45 is activated twice and G50 once that a rotatory table has to be measured.



N50003

N1 G17

N2 G54

N3 T1 M6

N4 M27

N5 G45 X-50 Y-20 Z0 C0 J1

N6 G45 X50 Y-20 Z0 J1

N7 G50 C1 N=54

N8 M28

N9 G54

N10 G0 Z100 C0

Set the plane of operation

Set the zero point

Load the touch trigger probe

Activate probe

Measure point 1

Measure point 2

Update of the zero offset value of the C-axis with the calculated angle

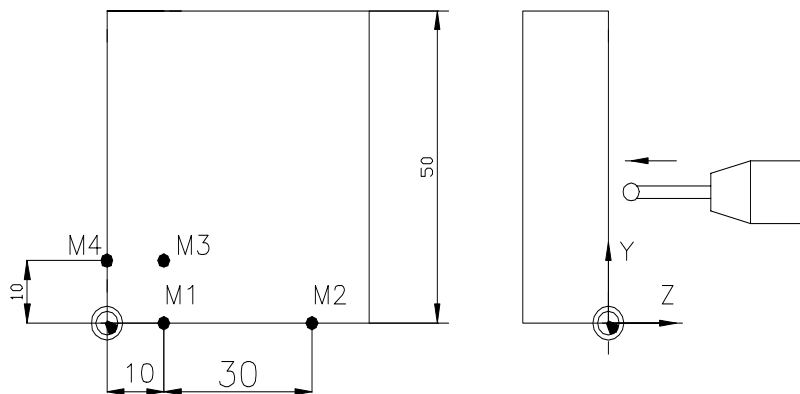
Deactivate probe

Set the zero point

Retract the tool and rotate the table to C0

Note: N7 G50 C1 N=54 C1=30 If in block N7 C1=30, then the table rotates 30 Grad extra so that the table is parallel to the X-Axis.

Example 4 Determining the zero point

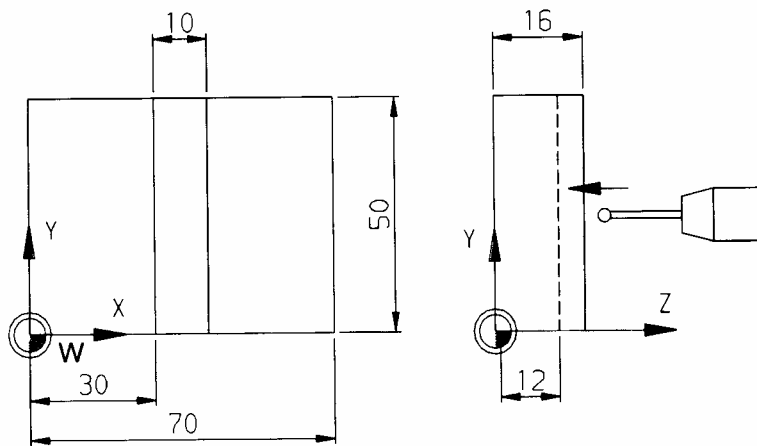


The probe is standing in the Z-axis. The part is mounted on a table rotating around the Z-axis. Five points of the part (M1 to M4 and again M1) are measured. M1 and M2 for covering the angular displacement; M3, M4 and M1 for measuring the positions of the axes.

The section of the partprogram for determination the zero point could be:

N50004	
N1 G54	Set the zero point
N2 G17	Set the plane of operation to be the XZ-plane
N3 G0 X10 Y-10 Z10 T1 M6	Load the touch trigger probe and move to the programmed position
N4 M27	Activate probe
N5 G45 X10 Y0 Z-5 C0 J1	Measure point M1
N6 G0 Z10	Retract the probe to avoid collision
N7 G45 X40 Y0 Z-5 J1	Measure point M2
N8 G0 Z10	Retract the probe to avoid collision
N9 G50 C0 N=54	Update of the zero offset value of the C-axis with the calculated angle
N10 G54	Set the zero point
N11 G0 C0	Rotate the table to C0.
N12 G45 X10 Y10 Z0 K-1	Measure point M3 to determine the position in the tool axis
N13 G0 Z10	Retract the probe to avoid collision
N14 G45 X0 Y10 Z-5 I1	Measure point M4 to determine the position in the X-axis
N15 G0 Z10	Retract the probe to avoid collision
N16 G45 X10 Y0 Z-5 J1	Measure point M1 to determine the position in the Z-axis
N17 G0 Z50	Retract the probe to avoid collision
N18 G50 X1 Y1 Z1 N=54	Update of the zero offset values of the X-, Y- and Z-axis
N19 G54	Set the updated zero point
N20 M28	Deactivate probe

Example 5 Correcting the length of a tool



With a mill a groove is made, the depth of the groove is measured and the tool length of the mill updated.

N90005

N1 G17

N2 T1 M6 (Mill radius 5 mm)

N3 X35 Y60 Z12 S1000 M3

N4 G1 Y-10 F200

N5 G0 Z200 M5

N6 T2 M6 (Probe)

N7 M27

N8 G45 X35 Y25 Z12 K-1

N9 G50 T1 L1=1

N10 M28

N11 Z200 M30

Set the plane of operation to be the XY-plane

Load the mill of 10 mm diameter

Start the spindle and move the mill to the start point of the groove

Mill the groove

Retract the tool and stop the spindle

Load the probe

Activate probe

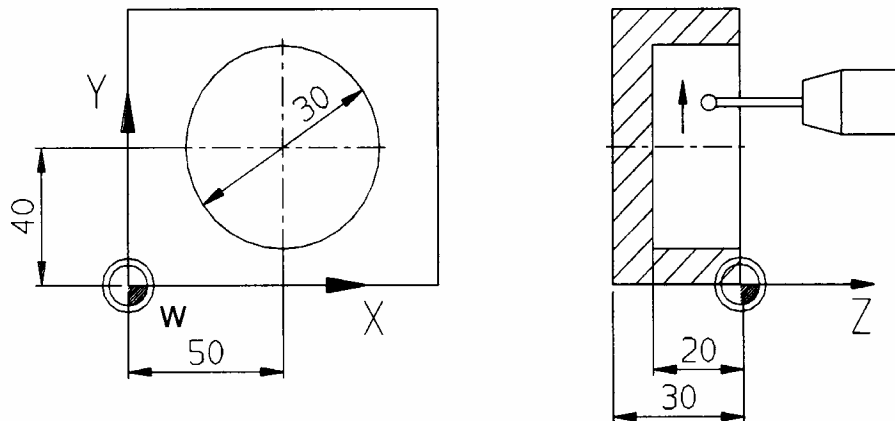
Measure the point in the negative direction of the tool axis

The calculated difference in the Z-axis is used to correct the length of tool 1

Deactivate probe

Retract the probe and end of program

Example 6 Milling and measuring a hole



A hole is milled and measured with a touch trigger probe. Checks are provided to see if the tolerance on the radius of the hole is within the required limits. If the radius is too small, the hole is milled again. If the radius is too large the part is rejected and a message displayed.

The partprogram could be:

N50006	
N1 G54	Set the zero point
N2 G17	Set the plane of operation to be the XY-plane
N3 T1 M6 (Mill radius 5 mm)	Load a mill with a diameter of 10 mm
N4 G89 Z-20 B2 R15 K6 F300 S1000 M3	Define the fixed cycle for milling the hole
N5 G79 X50 Y40 Z0	Mill the hole
N6 G0 Z50 M5	Retract the tool and stop the spindle
N7 T2 M6 (probe)	Load the touch trigger probe
N8 M19	Spindle stop at a certain angle
N9 M27	Activate probe
N10 G46 X50 Y40 Z-10 R15 I1 J1 F500 E5	Measure the hole at four points
N11 G0 Z50	Retract the probe to avoid collision
N12 G49 R.02 R1=2 N=19 E5	Check to see if the radius of the hole is not too big, (less than 15+.02). If the radius is too big, reject the part and display a message to it.
N13 G49 R2 R1=.02 N=15	Check to see if the radius of the hole is not too small, (greater than 15-.02). If the radius is too small, update the radius value in the tool memory and mill the hole again.
N14 G29 E1 E1=1 N=21	Jump to the end of the program
N15 G50 T1 R1=1	Update of tool radius in tool memory
N16 M28	Deactivated probe
N17 G14 N1=3 N2=6	Repeat of the blocks N3 to N6 to mill the hole within tolerance.
N18 G29 E1 E1=1 N=21	Jump to the end of the program
N19 M0 (HOLE OUT OF TOLERANCE)	Stop the program execution and display a message
N20 M30	End of program

5.29 G51/G52 Cancel/activate pallet zero point shift

Fix pallet zero offset with programmed value.

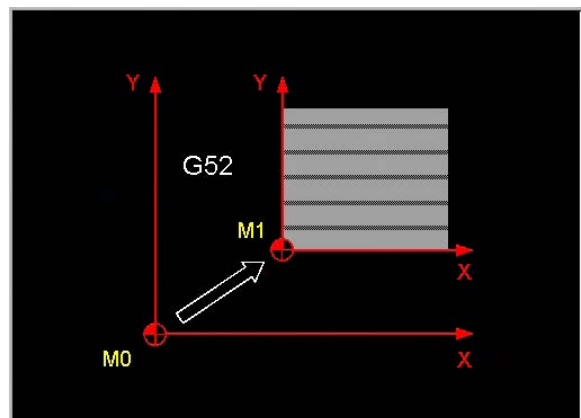
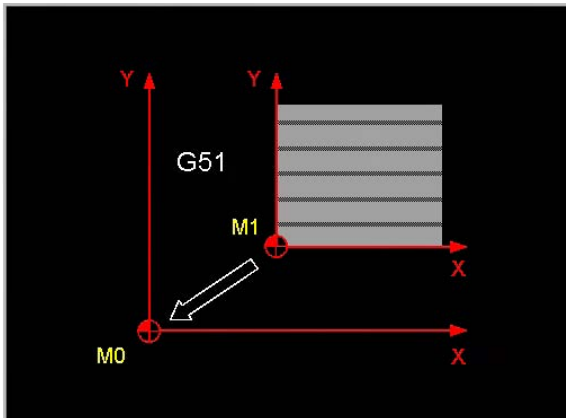
Format

Activate and deactivate after G51 has been used:

G52

To cancel:

G51



Notes and usage

Modality

G51 and G52 are modal functions.

Cancelling

The function G52 can be cancelled with softkey CLEAR CONTROL or overridden by programming G51.

The functions G51 and G52 will stay active after Softkey CANCEL PROGRAM, M30 or turning off the controller.

Other zero point shifts

If there is a zero point shift G54.[nr] active. Then G52 is active with the zero offset. If G52 is active and G54.[nr] are activated then G54.[nr] are activated with the zero offset of G52.

Function G52 is used for the purpose of automation, for instance pallet control. In this case the values for G52 are set by an IPLC program.

When MC84 = 0 then G52 is stored in ZO.ZO (zero point).

When MC84 > 0 then G52 is stored in PO.PO (Pallet Offset).

In ZO.ZO and in PO.PO the zero points can be edited.

5.30 G53/G54—G59 Cancel/activete zero point shift

On two ways the zero point shift can be done:

- 1) MC84=0 with G53/G54—G59
- 2) MC84>0 with G53 and G54.<Nr.>

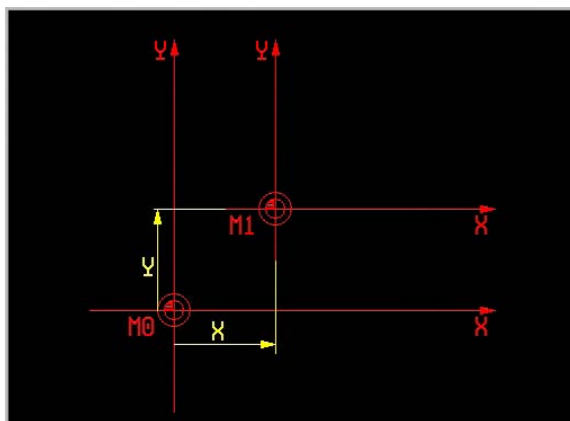
Apart from the present zero point shift table G54..G59 there is another zero point shift table G54 I[nr] showing a maximum of 99 zero point shifts. The appropriate zero point shift is selected by machine constant MC84.

The functionality is the same as that of the present zero point shift memory G54..G59, except for the following extensions and differences:

- 99 potential zero point shifts in the zero point shift memory
- Identification of zero point memory Ze.Ze (MC84 > 0)
- Programming (shift values) of zero point shift in NC program
- Programming of angle of rotation (B4=) in zero point shift
- Programming of zero point shift with an index (G54 I[nr])
- Comment is entered in zero point shift memory

5.30.1 G53/G54—G59 Cancel/activate zero point shift (MC84=0)

To shift the workpiece zero point to a new position whose coordinate values are stored in the zero point memory (under the appropriate number).



G	Activate zero point shift
X	Zero point coordinate
Y	Zero point coordinate
Z	Zero point coordinate
A	Zero point angle
B	Zero point angle
C	Zero point angle

Only G54

Format

To activate:

G54 {X..} {Y..} {Z..} {A..} {B..} {C..}
G55, G56, G57, G58 or G59

To cancel:

G53

Notes and Usage

Modality

G53 and G59 are modal functions.

Associated functions

G51/G52, G92, G93

Machine zero points

If a machine has several clamping stations or more than one rotary table it is necessary to state secondary machine zero points. These points are related to the geometric machine zero point (M_0). The axial distances measured from M_0 specify the position of these secondary zero points and are stored in the Zero Offset Memory together with their identifying G-function.

Entering in zero point shift memory

Shift values can be entered in the zero point memory in two different ways:

- The values of zero point shifts G54 – G59 are entered into the zero point shift memory via the control panel or through a data carrier before the program is executed.
- The values of zero point shift G54 X.. Y.. Z.. A.. B.. C.. are programmed in an NC program block. When the program is edited, the programmed values are accepted in the zero point shift memory and activated.

G52 zero point shift

The function G52 is not influenced by one of the functions G53 to G59.

Absolute /Incremental zero point shifts G92/G93

A programmed zero point shift (G92 or G93) is cancelled by one of the functions G53 to G59.

Scaling, mirror image and axis rotation (G73, G92/G93)

One of the functions G53 to G59 can be used in a program sequence, which is scaled, mirrored or rotated. The zero point shift is performed in the coordinate system of the machine tool and not influenced by the programmed change of coordinates.

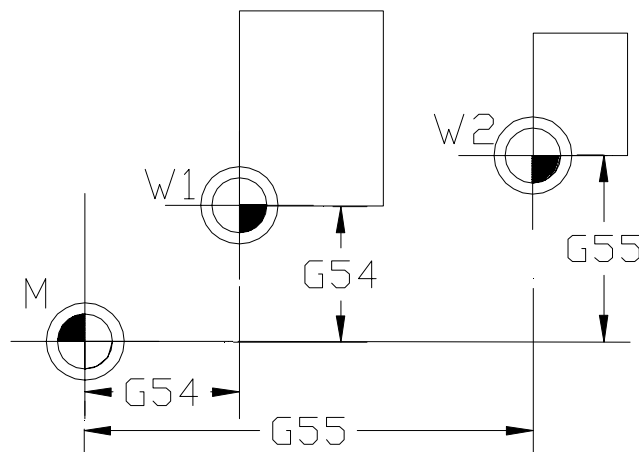
Cancellation

A secondary machine zero point can be overridden by programming G53.

G53 is automatically set at switching on the control and after a reference point search.

The functions G54 to G59 are not cancelled by CLEAR CONTROL, M30 or Softkey CANCEL PROGRAM.

Example



N60 G54

Select the secondary machine zero point W1. Its coordinates (X40, Y100, Z300) are retrieved from the Zero Offset Memory. All programmed coordinates are measured from W1.

N600 G55

Select the secondary machine zero point W2. Its coordinates (X200, Y100, Z100) are retrieved from the Zero Offset Memory.

Machine zero point W1 is cancelled and W2 is now active, therefore all programmed coordinates are measured from W2.

N700 G53

Cancel with G53 machine zero point W2. Then the coordinates (X0, Y0, Z0) are cleaned.

Zero point W2 will be deleted and M (machine zero point) will be active. After this action all coordinates are measured from M.

5.30.2 G54 Extended zero point shift (MC84>0)

To shift the workpiece zero point to a new position. The coordinate values can be entered in the zero point shift memory or programmed in the NC program block.

Format

Define and use zero point shift as follow:

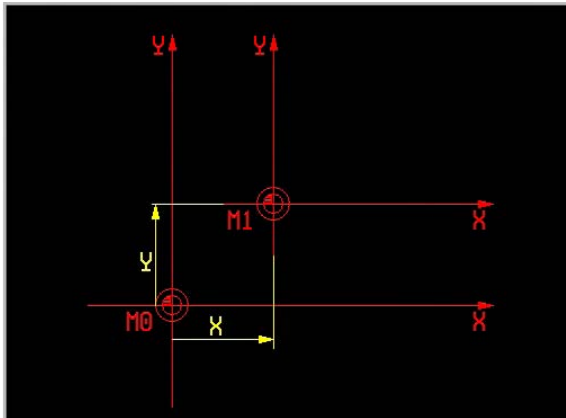
G54 I[nr] [Axis coordinates] {B4=..}

Use zero point shift as follow:

G54 I[nr]

To cancel

G53



G	Activate zero point shift
X	Zero point coordinate
Y	Zero point coordinate
Z	Zero point coordinate
A	Zero point angle
B	Zero point angle
C	Zero point angle
I	Zero point index
B4=	Angle of rotation absolute

A zero point shift can contain up to 6 Axis coordinates.

Notes and usage

Modality

G53 and G59 are modal functions.

Associated functions

G50, G51, G52, G53, G54 ... G59, G92, G93, G149, G150

Number of zero points

The number of possible zero point shifts in the table is determined by a machine constant (MC84). (0 < MC84 < 99).

Changing machine constant MC84

The zero point shift table is adjusted in the event of scaling (MC84 > 0). The existing zero points is maintained. Extended zero points are initialised to zero.

Attention:

If MC84 is zeroed, the table is changed (ZE.ZE is changed to ZO.ZO). The new zero point table is initialised to zero.

Entering in zero point shift memory

Shift values can be entered in the zero point memory in two different ways:

- The values of zero point shifts G54 I[nr] are entered into the zero point shift memory via the control panel or through a data carrier before the program is executed.
- The values of zero point shift G54 I[nr] X.. Y.. Z.. A.. B.. C.. B4=.. are programmed in an NC program block. When the program is edited, the programmed values are accepted in the zero point shift memory and activated.

Attention: If no new zero point shift values have been programmed in the program block, the zero point shift values already stored in the memory are not overwritten or deleted. The axis coordinates not programmed are taken from the memory. Risk of collision!

Comment

Additionally, each zero point shift in the table may be commented.

Axis rotation

Additionally, each zero point shift in the table may involve an axis rotation. First, the shift is executed and then the coordinate system is rotated through angle B4=.

Machine zero points

If a machine has several clamping stations or more than one rotary table it is necessary to state secondary machine zero points. These points are related to the geometric machine zero point (M0). The axial distances measured from M0 specify the position of these secondary zero points and are stored in the Zero Offset Memory together with their identifying G-function.

Zero point shift memory

All values of zero point shifts G54 I[nr] should be stored in the zero point shift memory via the control panel or by a data carrier, before the program is executed.

G52 zero point shift

G52 does not affect the functions G53...G59. If G52 is active, G54..G59 will be effective from this shift onwards.

Absolute / incremental zero point shifts (G92/G93)

A programmed zero point shift (G92 or G93) is deleted from any of the G54 I[nr] functions.

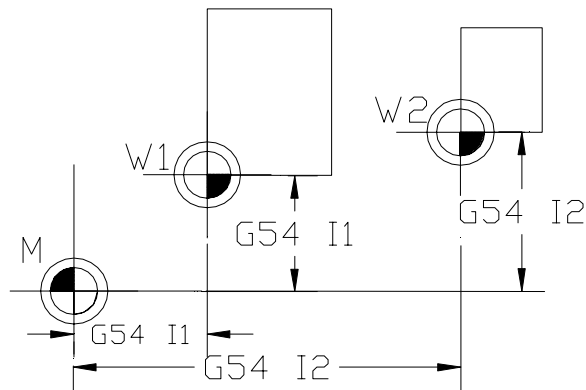
Scaling, mirroring and axis rotation (G73, G92/G93)

It is allowed to use any of the functions G54 I[nr] in a program section which is to be scaled, mirrored or rotated. The zero point shift takes place in the coordinate system of the machine tool and is not affected by the programmed change of coordinates.

Delete

G54 I[nr] is automatically deleted by the CLEAR CONTROL Softkey and by programming G53.

G54 I[Nr.] is not deleted when the CANCEL PROGRAM Softkey or M30 is used.

Examples**Example 1**

N60 G54 I1

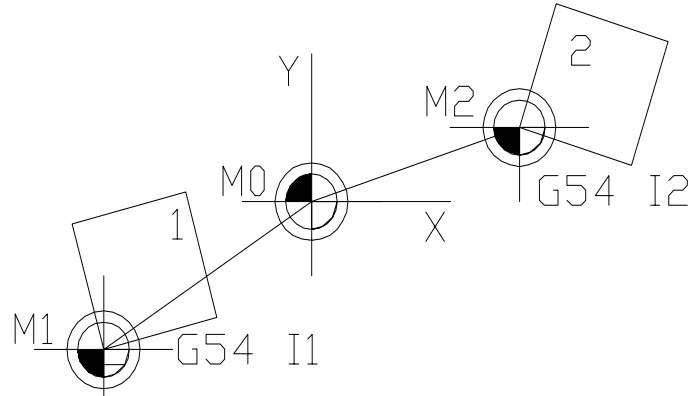
Select the secondary machine zero point W1. Its coordinates (X40, Y100, Z300) are retrieved from the Zero Offset Memory. All programmed coordinates are measured from W1.

N600 G54 I2

Select the secondary machine zero point W2. Its coordinates (X200, Y100, Z100) are retrieved from the Zero Offset Memory. Machine zero point W1 is cancelled and W2 is now active, therefore all programmed coordinates are measured from W2.

N700 G53

Cancel with G53 machine zero point W2. Then the coordinates (X0, Y0, Z0) are cleared. Zero point W2 will be deleted and M (machine zero point) will be active. After this action all coordinates are measured from M.

Example 2 **Axis rotation**

Entry in zero point table and calling:

N60 G54 I1 X-42 Y-15 B4=14 (Z0 C0)

Zero point shift values are entered in the zero point shift table.

Machine workpiece 1, all programmed coordinates are measured from M1.

N120 G54 I2 X10 Y24 B4=-17

Machine workpiece 2, all programmed coordinates are measured from M2.

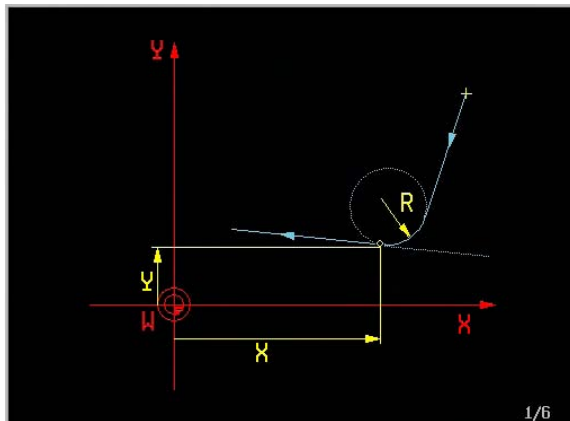
5.31 G61 Tangential approach

Programs a tangential approach movement between a starting point and start of a contour.

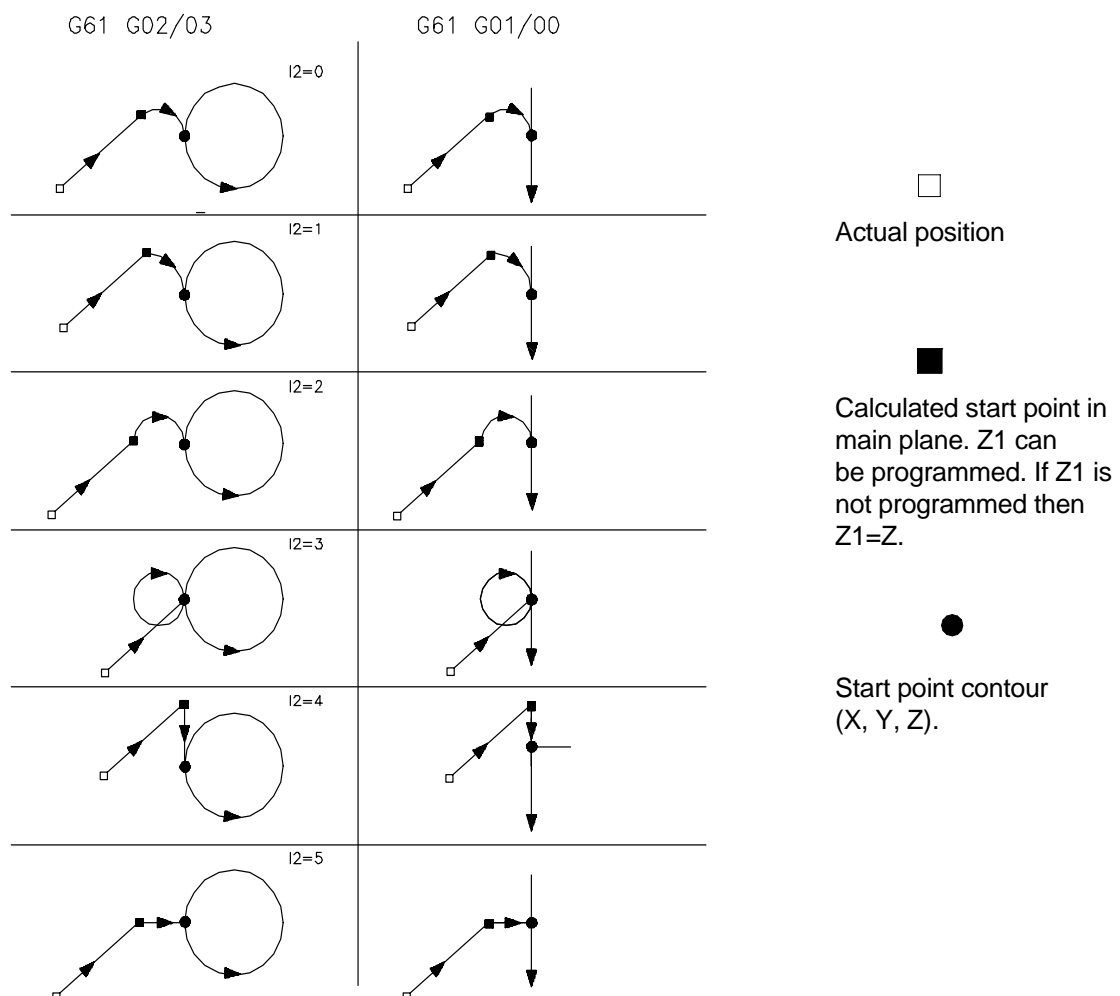
Format

G61 {I2=} X... Y... Z... R... {Z1= or Y1= or X1=} {I1=} {F2=}

G61 {I2=} B2=... L2=... Z... R... {Z1= or Y1= or X1=} {I1=} {F2=}



G Tangential approach
 X Endpoint tangential approach
 Y Endpoint tangential approach
 Z Endpoint tangential approach
 P Point definition number
 R Radius
 Z1= Startpoint in Z
 B2= Polar angle
 ?90= Endpoint abs. (X,Y,Z...)
 ?91= Endpoint incr. (X,Y,Z...)
 I1= Linear movement 0=rapid,1=feed
 I2= Tangential approach definition
 L2= Polar length



Z1=	Start height calculated begin point in Z
I1=	Linear movement 0=rapid, 1=feed.
I2=	Approach method
	I2=0 with line and tangent circle
	I2=1 with quarter circle
	I2=2 with semicircle
	I2=3 Helix for feeding (for pockets)
	I2=4 Parallel to contour
	I2=5 Vertical

Notes and usage

Startpoint of the approach movement

The Controller calculates the start point. The first movement is a positioning movement towards the calculated start point. From this position the approach movement towards the contour is started.

Height of the startpoint in tool-axis

The Z (G17) parameter contents the contour start point in the tool-axis. The parameter Z1 (G17) contents the height of the start point of the approach movement. By a difference of Z and Z1 a slanting movement is made. (In G18 Y and Y1=)

Approach movement

The approach movement consists of two movements. The first movement is a rapid or a feed movement (depends if I1=0 or I1=1) towards the calculated start point. The second movement is a feed movement to the starting point of the contour.

Approach side

The parameters G41 or G42 determine the approach side. (G41 is left, G42 is Right). When G40 is active the approach is the same as the in G41 (left side).

The circular arc movement

The circular movement is determined by the position of the starting point of the contour and the starting point of the tangential movement. The Direction of the tangential circular movement is determined by the direction of the contour movement. The movement is always fluid between the tangential circular arc and the contour movement.

Radius compensation

The radius compensation (G41/G42) must be activated right before G61 is active the compensation will be activated during the linear movement. The actual position determinates the calculated position. If the radius compensation is active the linear movement and the circular movement are operated according to the radius compensation.

Perpendicular approach movement

The position of the contour start point determines the position of the perpendicular approach movement.

G1-function

If there is no G-function programmed after a G17 Block G1 will not be active.

Limitation when I2=0 is active

When the actual position is further away then one diameter from the circular movement then the approach movement contains a linear and a circular movement. When the actual position is within the circular movement then parameter I2= changes from 0 to 1, and the begin movement contains a quarter circle movement.

Limitations

Programming a G61 has the following limitations:

- G61 is in G64 mode NOT allowed
- G61 is in MDI NOT allowed
- G61 is in G182-Mode NOT allowed

The program block after G61 has limitation. The following functions are allowed:

- G64
- G0, G1, G2, G3 with movements in the active plane
- Search to movement is done, when no movement is found.

Remark: From V410 the result of the G61, comparing with earlier versions, will be different.

Notes: The programmer must program the approach movement in such a way that during milling the contour doesn't damage.

Support programming

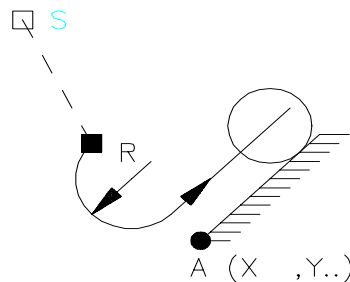
The functions "tangential approach" (G61) and "tangential exit" (G62) can be used in the following mode:

- free entry
- support entry

The SUPPORT ENTRY supports the programmer with pictures and text.

Examples

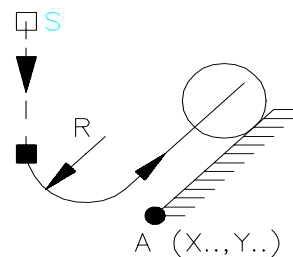
Example 1



I2=2

```
N33 G17
N34 G0 X... Y... (S)
N35 G41
N36 G61 I2=, X... Y... I1=0 R5
```

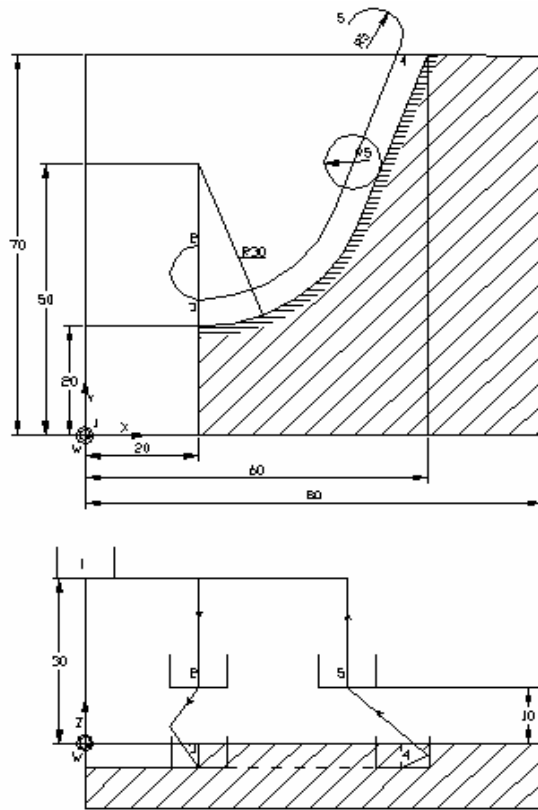
```
N37 G64
```



I2=0

Define the main plane. All the movement occur in one plane.
Moving to the start point position.
Radius compensation must be active for the left side.
Tangential approach movement. This movement contains two movements. First a linear movement to the calculated position (with rapid (I1=0) or feed (I1=1)) and then a circular movement.
Start a contour.

Example 2



N1 G17
 N2 T1 M6 (Mill R5)
 N3 F500 S1000 M3
 N4 G0 X0 Y0 Z50

N5 G41
 N6 G61 I2=2 X20 Y20 Z-5 Z1=10 R5 I1=0 F2=200

N7 G64
 N8 G3 I20 J50 R1=0
 N9 G1 X60 Y60
 N10 G63

Active XY-plane (G17).
 Load Tool T1.
 Activating feed, speed and spindle rotation (M3).
 Move tool rapidly to programmed position (position 1: X0 Y0 Z30).
 Set radius compensation LEFT
 N6 G61 I2=2 X20 Y20 Z-5 Z1=10 R5 I1=0 F2=200

- Tangential approach movement
- I2=2 is a semicircle
- The first section is a rapid movement with positioning logic to the start point of the semicircle movement (position 2: X.. Y.. Z10). The Radius compensation is activated in this movement.
- The circular movement will perform a helix movement.. The contour starts with position X20 Y20 Z-5. (Position 3: X20 Y25 Z-5).

N7 G64
 Start the contour description.
 N8 G3 I20 J50 R1=0
 A circular movement tangential to a Line.
 N9 G1 X60 Y60
 Tangential Linear movement.
 N10 G63
 End of the contour description.

5.32 G62 Tangential exit

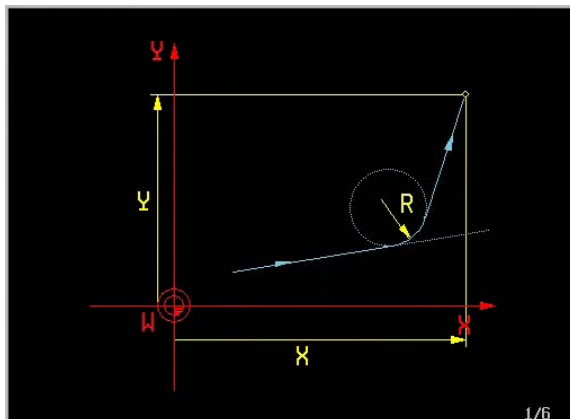
Programs a tangential exit after the end of the contour.

Format

G62 I2>0 Z1=... R... {I1=} {F2=}

G62 I2=0 X... Y... Z... Z1=... R... {I1=} {F2=}

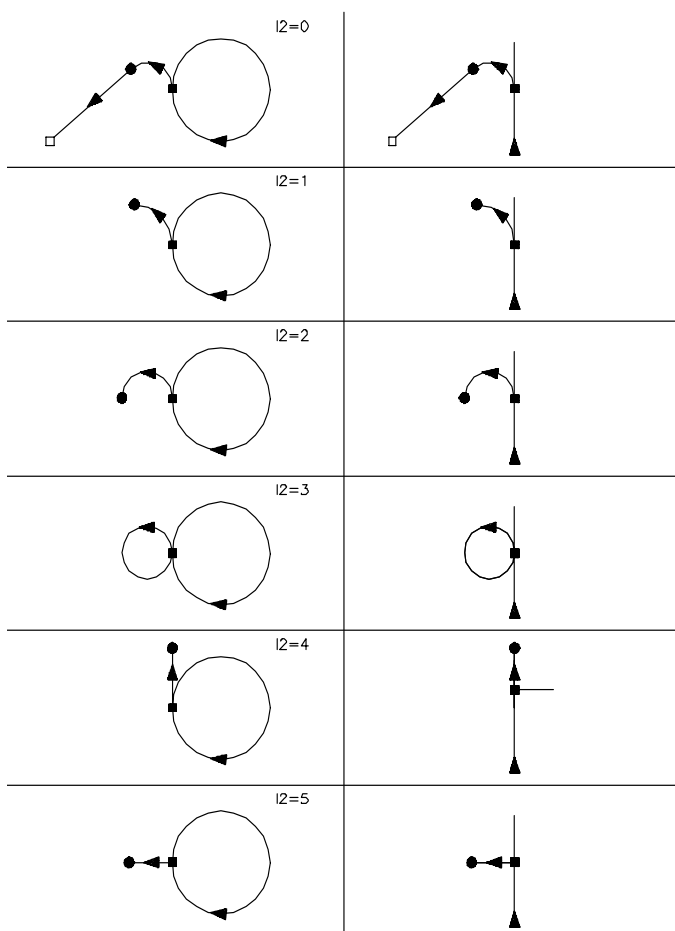
G62 I2=0 B2=... L2=... Z... R... {I1=} {F2=}



G Tangential exit
 X Endpoint tangential exit
 Y Endpoint tangential exit
 Z Endpoint tangential exit
 P Point definition number
 R Radius
 Z1= Startpoint in Z
 B2= Polar angle
 ?90= Endpoint abs. (X,Y,Z..)
 ?91= Endpoint incr. (X,Y,Z..)
 I1= Linear movement 0=rapid,1=feed
 I2= Tangential exit definition
 L2= Polar length

G62 G02/03

G62 G01/00



End point contour.

Calculated end point in Main plane. Z1 can be programmed. If Z1 is not programmed then the value doesn't change.

Programmed end point tangential exit (X, Y, Z) (only I2=0).

Z1= End height tangential exit
 I1= Linear movement 0=rapid, 1=feed
 I2= Tangential exit definition
 I2=0 with circle with tangent line.
 I2=1 with quarter circle
 I2=2 with semicircle
 I2=3 Helix for feeding (for pockets)
 I2=4 Parallel to contour
 I2=5 Vertical

Note: To understand the function G62 read first function G61.

Notes and usage

Programming rule

Just for the G62 a movement must programmed.

Cancel radius compensation (G40)

The radius compensation is deactivated in the G62 line. The movement to the calculated position will be done with radius compensation.

Remark: From V410 the radius compensation will be deactivating in G62.

Remark: From V410 the result of the G62, comparing with earlier versions, will be different.

Limitations

Programming a G62 has the following limitations:

- G62 is in G64 mode NOT allowed
- G62 is in MDI NOT allowed
- G62 is in G182-Mode NOT allowed

The program block after G61 has limitation. The following functions are allowed:

- G64
- G0, G1, G2, G3 with movements in de active plane

Support programming

The functions "tangential approach" (G61) and "tangential exit" (G62) can be used in the following mode:

- free entry
- support entry

The SUPPORT ENTRY supports the programmer with pictures and text.

G1-function

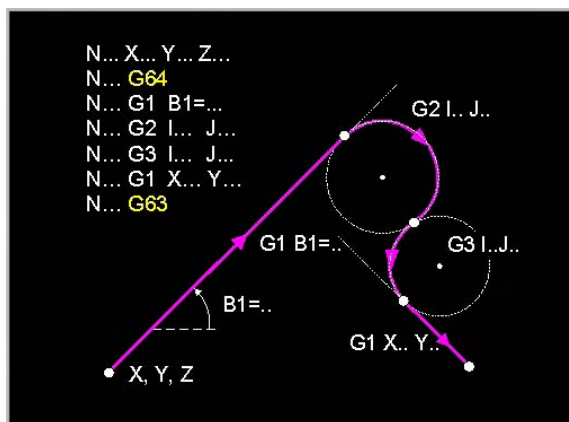
If there is no G-function programmed after a G17 Block G1 will not be active.

N1 G17	Active XY-plane (G17).
N2 T1 M6 (Mill R5)	Load Tool T1.
N3 F500 S1000 M3	Activating feed, speed and spindle rotation (M3).
N4 G0 X0 Y0 Z50	Move tool rapidly to programmed position (position 1: X0 Y0 Z30).
N5 G41	Set radius compensation LEFT
N6 G61 I2=2 X20 Y20 Z-5 Z1=10 R5 I1=0 F2=200	<ul style="list-style-type: none"> - Tangential approach movement - I2=2 is a semicircle - The first section is a rapid movement with positioning logic to the start point of the semicircle movement (position 2: X.. Y.. Z10). The Radius compensation is activated in this movement. - The circular movement will perform a helix movement. The contour starts with position X20 Y20 Z-5. (Position 3: X20 Y25 Z-5).
N7 G64	Start the contour description.
N8 G3 I20 J50 R1=0	A circular movement tangential to a Line.
N9 G1 X60 Y60	Tangential Linear movement with endpoint (position 4: x.. Y.. Z-5).
N10 G63	End of the contour description.
N11 G62 I2=2 Z1=20 R5	<ul style="list-style-type: none"> - Tangential Exit - I2=2 is a semicircle - The circular movement will perform a helix movement. Start point in the Z-axis is -5, and the endpoint is 10.
	Radius compensation will be deactivated.
N12 G0 X0 Y0 Z50	Retract Tool (position 1: X0 Y0 Z30)
N13 M30	Program End.

5.33 G63/G64 Cancel/Activate geometric calculations

G63: To cancel the geometric calculations and to return to programming complete blocks.

G64: To activate the geometric calculations.



G Activate geometric calculations

General principles for using the geometry

Between the functions G64 and G63 a contour can be described. An easy way of programming linear and circular movements makes it possible to let the control perform the necessary calculations for e.g. an intersection point or point of tangency. Each time a calculation is required at least two blocks of data are used. Each block is programmed with the standard G-functions for linear (G0 and G1) and circular movements (G2 and G3) and some information to define the line or circle. These blocks do not necessarily contain all data as previously specified, but with some special words (indicators) is achieved, that the missing data can be calculated by the control.

The first block establishes where the start point is located and what type of end point is required. The second block supplies the data for calculating the end point coordinates of the first block as e.g. a point of tangency or an intersection point of two elements. This end point is also the start point of the second block.

Between these movements can be inserted:

- A chamfer (between linear movements),
- A rounding (between intersecting elements),
- A connecting circle (between tangent elements or elements which do not meet)

It may happen that the second block does not supply enough data for calculating the end point of the first block. In that case the control looks for the next block and try to calculate the end point of the second block and first one. Up to five blocks are looking for in advance.

Format

G64 Activating geometric calculations

G0, G1, G2 or G3 Linear (G0/G1) and circular movements (G2/G3)

G63 Cancelling geometric calculations

Only the most commonly used formats are given here. Refer to a special appendix at the end of this manual for a detailed description of the possible formats for G0/G1 and G2/G3 in the many cases and also for examples of the use of the geometry.

For all formats the G64-function is assumed to have already been programmed in a previous block and is therefore active.

The XY-plane is also assumed to be the active plane. Refer to Notes and usage PLANE SELECTION for changes to be made in the formats, if another plane is active.

Possible parameters between G64 and G63 lines.**Straight line**

```

G   Linear interpolation
X   Endpoint coordinate
Y   Endpoint coordinate
Z   Endpoint coordinate
I   Chamfer length
X1= Arbitrary endpoint coordinate
Y1= Arbitrary endpoint coordinate
B1= Angle
B2= Polar angle
I1= Parallel shift
J1= 1=intersection left, 2=right
L2= Polar length
P1= Point definition number
R1= R1=0 tangent to line

```

Circle

```

G   Circular clockwise
X   Endpoint coordinate
Y   Endpoint coordinate
Z   Endpoint coordinate
I   Center point in X / pitch in X
J   Center point in Y / pitch in Y
K   Center point in Z / pitch in Z
R   Circle radius
B1= Angle
B2= Polar angle
B3= Polar angle for center
B5= Angle of arc
J1= 1=intersection left, 2=right
L2= Polar length
L3= Polar length for center

```

```

P1= Point definition number
R1= R1=0 tangent to line

```

Notes and Usage**Modality**

G64 is modal with G63

Cancellation

The geometric calculations are cancelled with the function G63. Thereafter complete blocks have to be programmed. In the last block before the cancellation of the geometric calculations an absolute position must be programmed.

Default mode

At CLEAR CONTROL the function G63 is automatically activated.

Permitted functions

G-functions allowed when the G64-function is active:

G0/G1/G2/G3; G4; G40/G41/G42/G43/G44; G94/G95

Functions **not allowed** when G64 is active

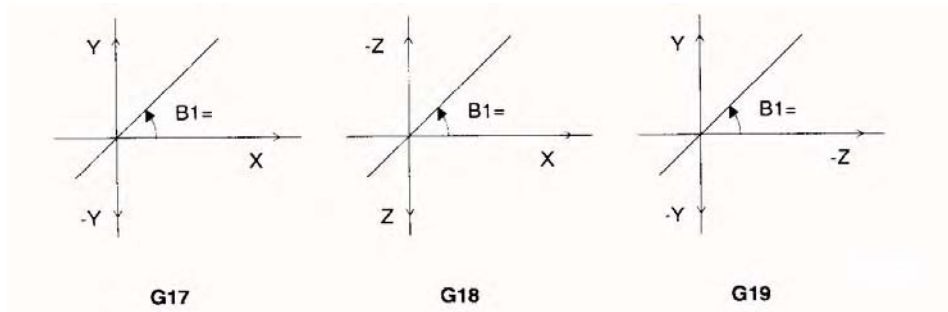
- All G-functions not mentioned in the table above
- Incremental programming (Cartesian and polar)
- Helix interpolation
- More than one defined point in a block
- The M-functions M6, M66 and M67

Plane selection

The geometric calculations are executed in the plane defined by G17 (XY- plane), G18 (XZ-plane) or G19 (YZ-plane).

In the three planes the angle B1= is defined with respect to:

- the + X-axis in the XY- or XZ-plane
- the - Z-axis in the YZ-plane



Angle definition in the different planes

A support point can be programmed with:

- X1= and Z1= in the XZ-plane
- Y1= and Z1= in the YZ-plane

Using macros

The use of the geometric calculations is allowed in a macro. All geometry blocks including G63 and G64 must be in the same macro.

Using repeat functions

The use of the geometric calculations is allowed in a section of a partprogram repeated by a G14 or G29. All geometry blocks including G63 and G64 must be in the same section to be repeated.

Scaling, mirror image and axes rotation

First activating scaling, mirror image or axes rotation and then using the geometric calculations is allowed and results in the required operation on the program section.

Dwell time

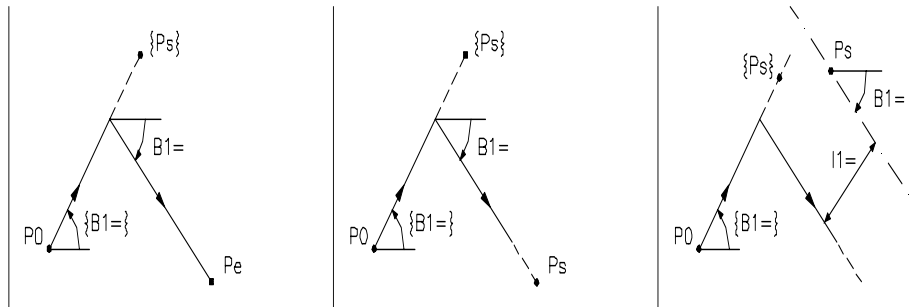
Within function G64 it is not possible to active G4.

Explanation of the possible formats:

In the illustrations in which the formats are explained, the following conventions are used:

- P₀= a start point known from the previous block
- P_s = a support point on a line or on a parallel line
- P_e = a programmed end point
- M = a programmed circle centre point
- R = a programmed radius of a circle

5.33.1 Intersection point between two straight lines



Intersection point with known start point from first line

Possible definition of the first line

N.. G1 {B1=..}

(Start point and angle)

N.. G1 X{Ps} Y{Ps}

(Start point and support point)

Possible definition of the second line

N.. G1 B1=.. X{Pe} Y{Pe}

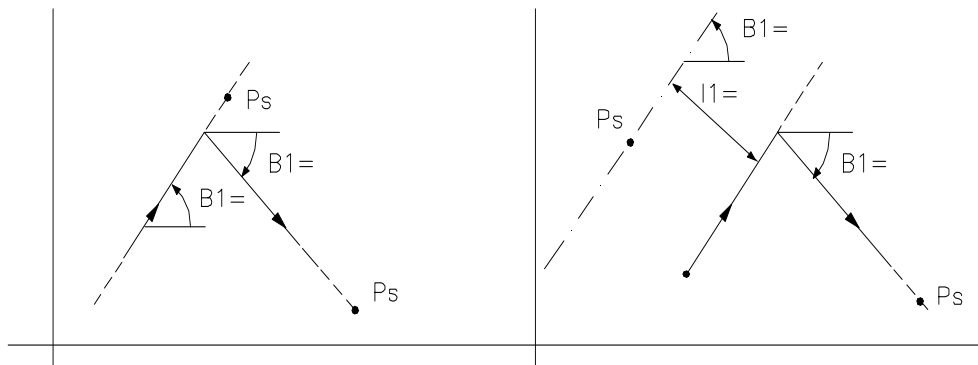
(Angle and end point)

N.. G1 B1=.. X{Ps} Y{Ps}

(Angle and support point)

N.. G1 X{Ps} Y{Ps} B1= I1=

(Support point, angle and parallel line)



Intersection point with unknown start point from first line

Possible definition of the first line

N.. G1 {B1=} X{Ps} Y{PS}

(Support point and angle)

N.. G1 B1= X{Ps} Y{Ps} I1=

(Support point, angle and parallel line)

Possible definition of the second line

N.. G1 B1=.. X{Pe} Y{Pe}

(Angle and end point)

N.. G1 B1=.. X{Ps} Y{Ps}

(Angle and support point)

N.. G1 X{Ps} Y{Ps} B1= I1=

(Support point, angle and parallel line)

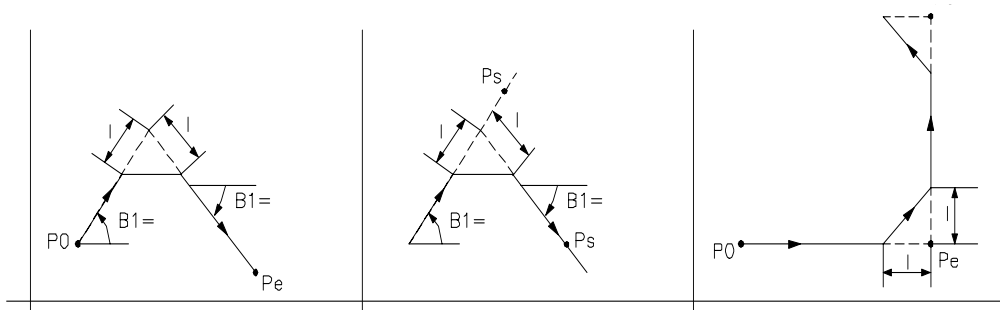
Note: It is also possible to program the intersection point as an end point, if it is known from the drawing. Refer to END POINT in Notes and usage for details.

A chamfer inserted between two intersecting lines

The lines are programmed as indicated in the previous section.

The chamfer is:

- symmetrically located around the intersection point
- programmed with G1 and the length of the chamfer (I-word)



Chamfer between two straight lines

N.. G1 {B1=..} or {support point/parallel line}
 N.. I.. {chamfer parameter}
 N.. B1=.. {end point} or {support point/parallel line}

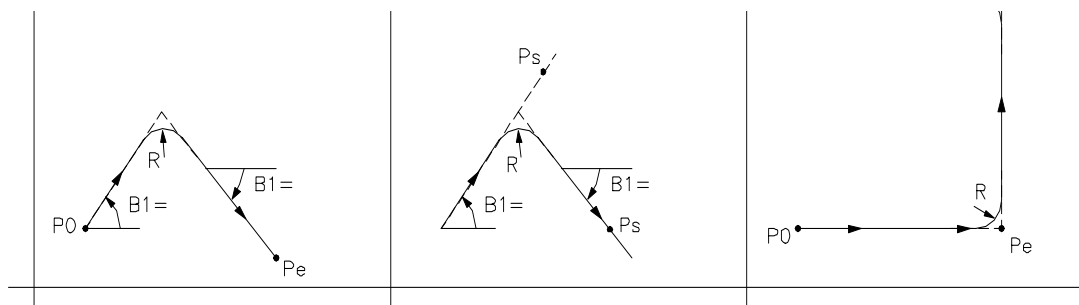
A rounding between two intersecting lines

The lines are programmed as indicated in the previous section.

The rounding is tangent to the line from the previous block and to the line of the next one.

A rounding is programmed with:

- G2 or G3 indicating the direction of movement,
- the radius (R-word) of the rounding.



Rounding between two straight lines

N.. G1 {B1=..} or {support point/parallel line}
 N.. G2/G3 R.. rounding
 N.. G1 B1=.. {end point} or {support point/parallel line}

Note: It is also possible to insert a rounding between a straight line and a chamfer or between a chamfer and a straight line.

N.. G1 {B1=..} or {support point/parallel line}
 N.. G2/G3 R.. rounding
 N.. I.. {chamfer parameter}
 N.. G2/G3 R.. rounding
 N.. G1 B1=.. {end point} or {support point/parallel line}

5.33.2 Intersection point indicator

If an intersection point between line and circle or two circles should be calculated, two points are possible.

With the word J1= is indicated which point is required:

- J1=1: the left intersection point (P1)
- J1=2: the right intersection point (P2).

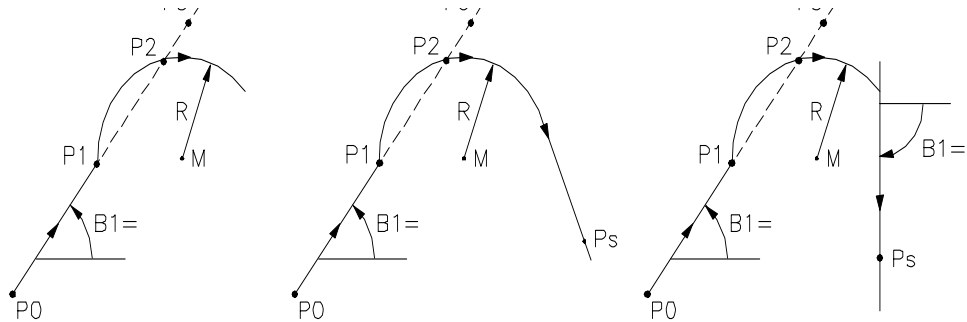
With a line through the circle centre point

- J1=1: the smallest distance between the start point or end point
- J1=2: the largest distance between the start point or end point.

With a line that starts or ends in the circle centre.

- J1=1: is the first intersection.
- J1=2: is the second intersection.

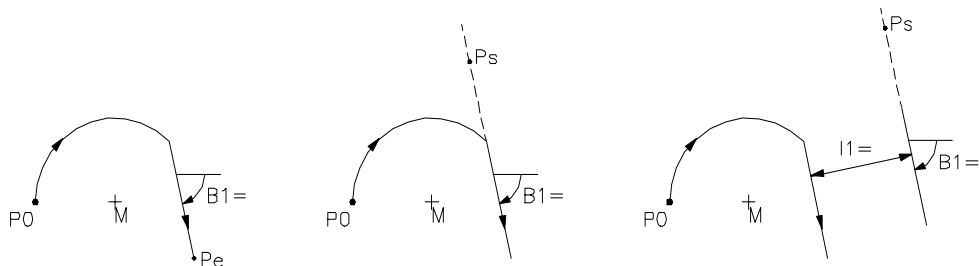
5.33.2.1 Intersection point between line and circle or circle and line



Line to circle

N..G1{B1=..} or {support point} J1=1/2

N..G2/G3 I..J.. R..

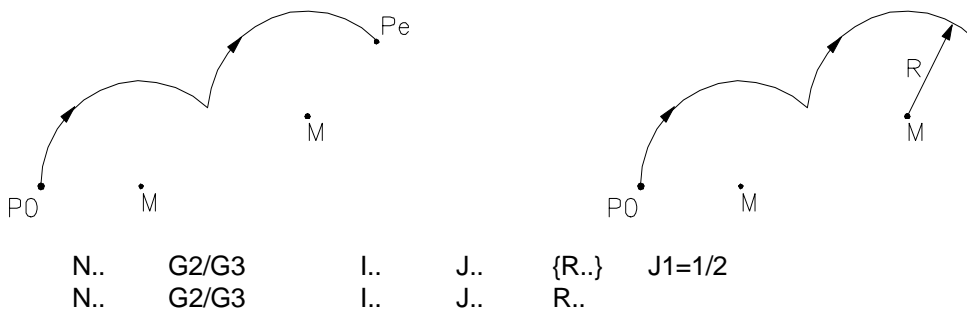


Circle to line

N.. G2/G3 I.. J.. R.. J1=1/2

N..G1 B1=..{support point} or {end point}

5.33.2.2 Intersection point between two circles



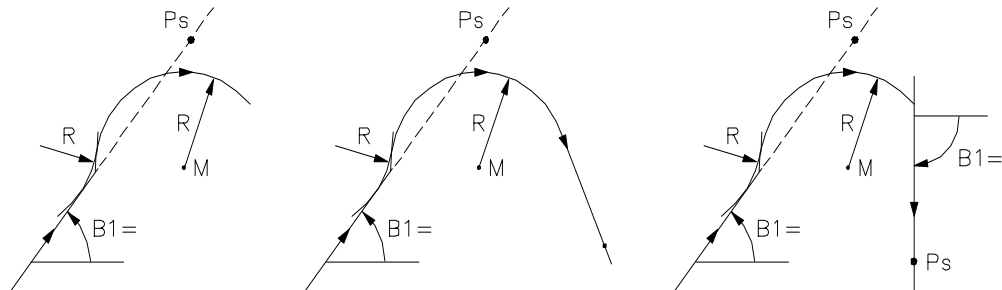
N.. G2/G3 I.. J.. {R..} J1=1/2
N.. G2/G3 I.. J.. R..

5.33.3 Programming a rounding

A rounding is always tangent to the geometry elements (line or circle) from the previous block and the next one. A rounding is programmed with:

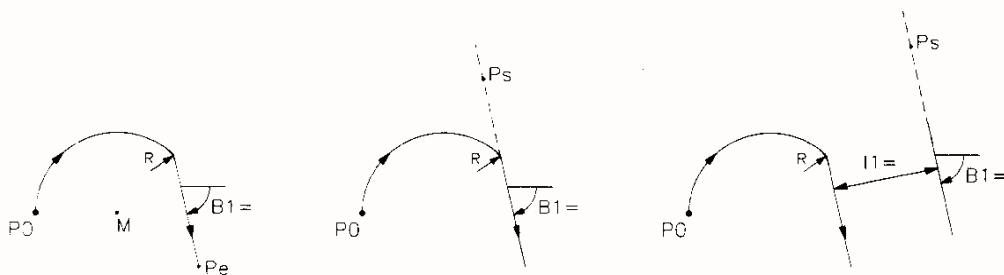
- G2 or G3 indicating the direction of movement,
- the radius (R-word) of the rounding.

5.33.3.1 A rounding between intersecting line - circle or circle - line



Line to circle

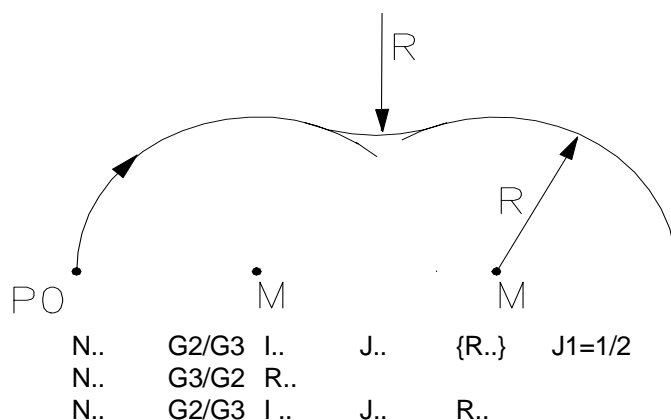
N.. G1 {B1=..} or {support point} J1=1/2
 N.. G2/G3 R..
 N.. G3/G2 I.. J... R..



Circle to line

N.. G3/G2 I..... J... {R..} J1=1/2
 N.. G2/G3 R..
 N.. G1 B1=.. {support point} or {endpoint}

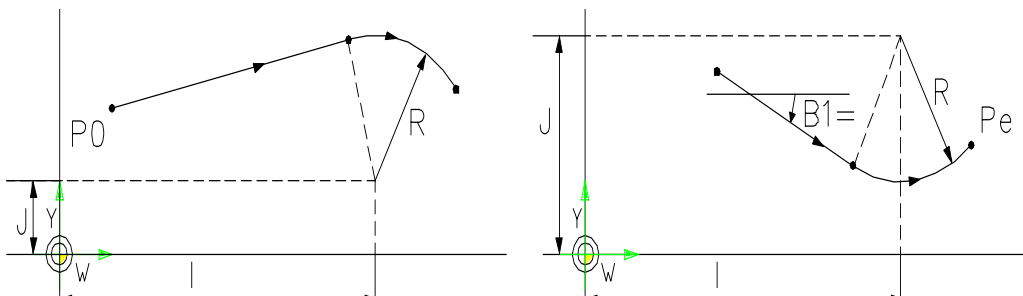
5.33.3.2 A rounding between two intersecting circles



5.33.4 Two tangent geometry elements

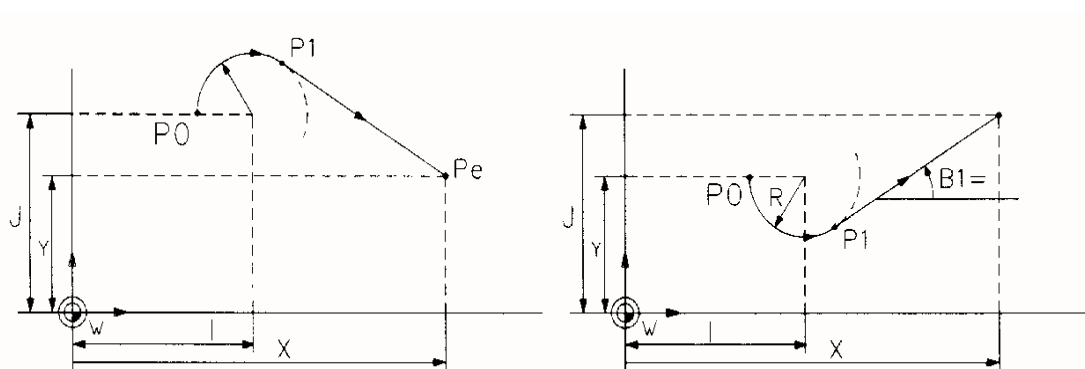
5.33.4.1 Tangency indicator

With the word R1=0 in the first block is indicated, that a line is tangent to a circle or a circle tangent to a line or another circle.



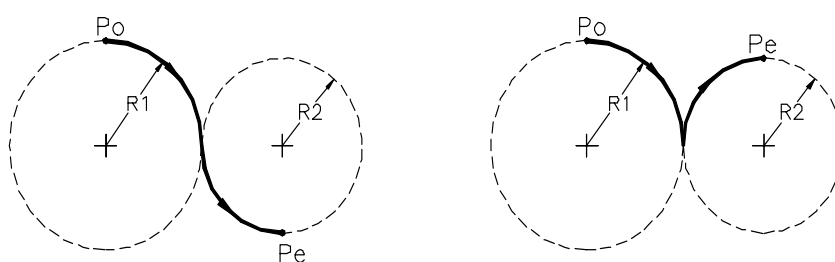
Line tangent to circle

N.. G1 {B1=..} or {support point} R1=0
N.. G2/G3 I.. J.. R..



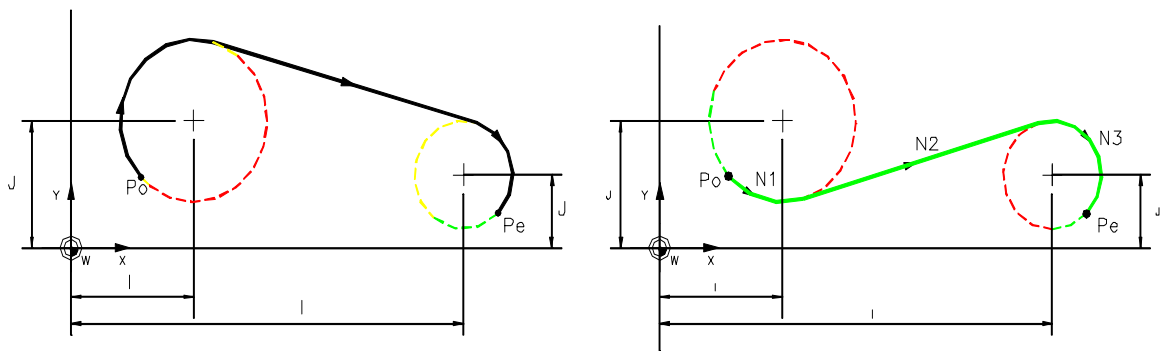
Circle tangent to line

N.. G2/G3 I.. J.. (R..) R1=0
N.. G1 {B1=..} or {support point} or {end point}



Two tangent circles

N.. G2/G3 I.. J.. {R..} R1=0
N.. G2/G3 I.. J.. R..



Common tangent line from two circles

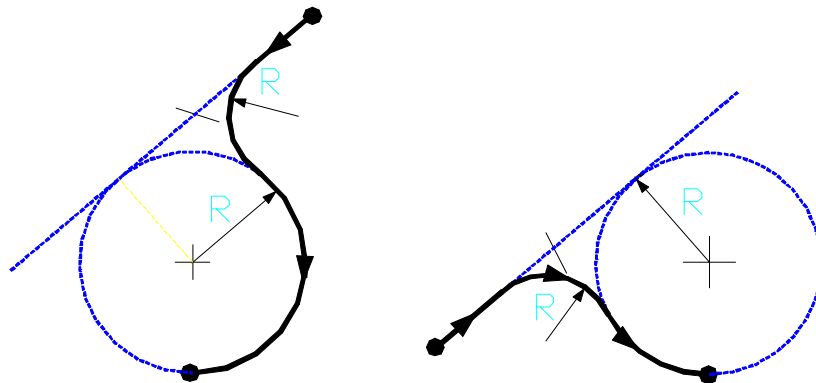
N..	G2/G3	I..	J..	{R..}	R1=0
N..	G1				R1=0
N..	G2/G3	I..	J..	R..	

5.33.5 Connecting circles

A connecting circle is:

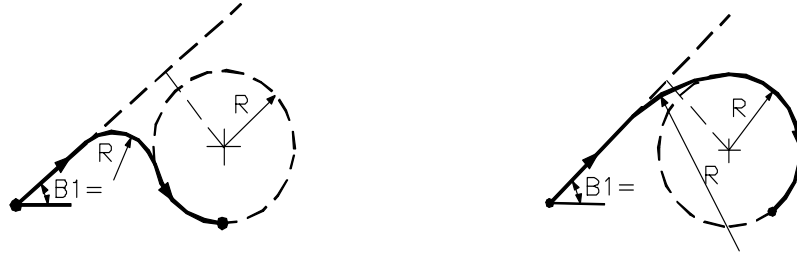
- always tangent to the geometry elements (line or circle) from the previous block and the next one.
- programmed with:
- G2 or G3 indicating the direction of movement,
- the radius (R-word) of the circle.

5.33.5.1 A connecting circle between line and circle or circle and line



Line tangent to circle

N..	G1	{B1=..}	or	{support point}	R1=0
N..	G3/G2	R..			
N..	G2/G3	I..	J..	R..	

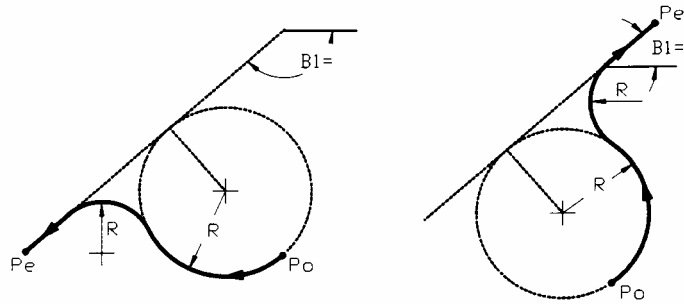


Line does not meet a circle

N.. G1 {B1=..} or {support point}

N.. G3/G2 R..

N.. G2/G3 I.. J.. R..

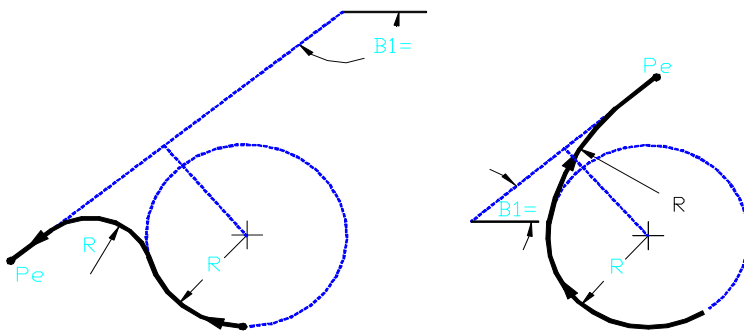


Circle tangent to line

N.. G2/G3 I.. J.. R.. R1=0

N.. G3/G2 R..

N.. G1 {B1=..} or {support point} or {end point}



Circle does not meet the line

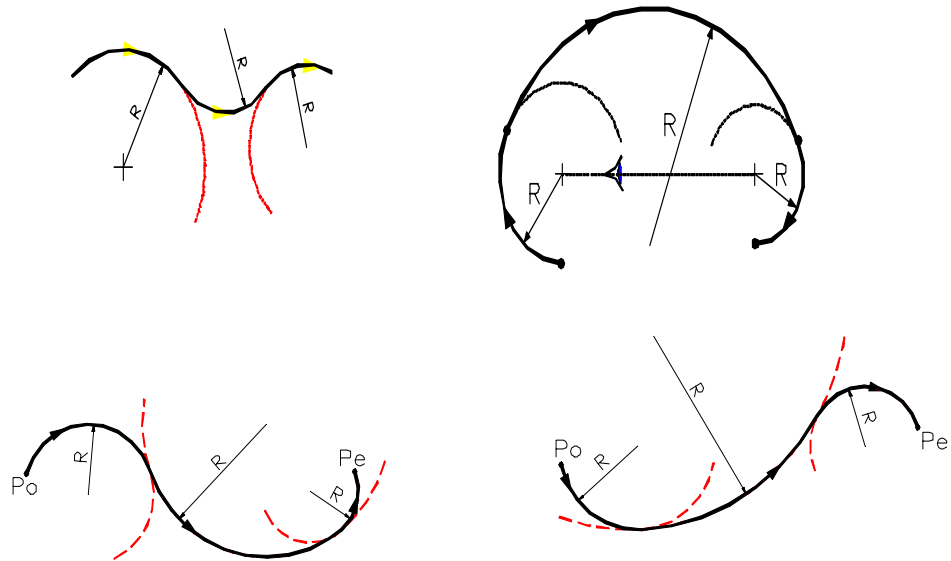
N.. G2/G3 I.. J.. R..

N.. G3/G2 R..

N.. G1 {B1=..} or {support point} or {end point}

5.33.5.2 A connecting circle between two circles outside each other

To insert a connecting circle between two circles outside each other which do not meet. The direction of rotation on the three circles indicates the type of connecting circle.

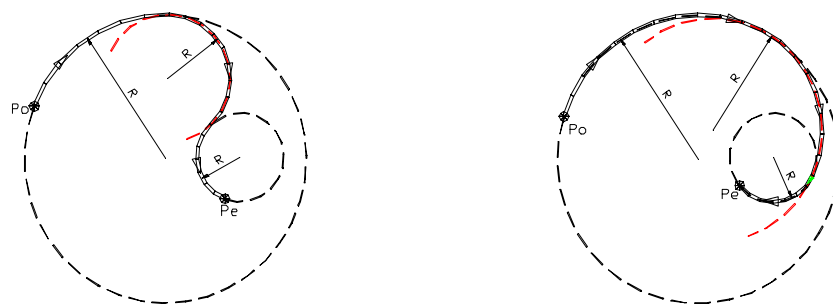


For all cases the same format is available:

N..	G2/G3	I..	J..	{R..}
N..	G3/G2	R..		
N..	G2/G3	I..	J..	R..

5.33.5.3 A connecting circle between two circles of which one circle inside the other one

To insert a connecting circle between a circle inside the other one which do not meet. The direction of rotation on the three circles indicates the type of connecting circle.



For both cases the same format is available:

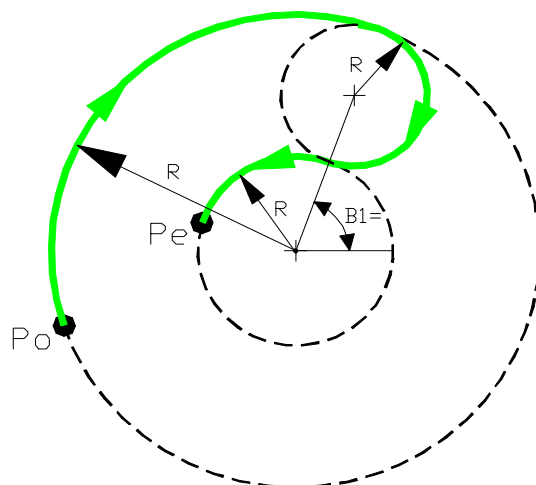
N..	G2/G3	I..	J..	{R..}
N..	G2/G3	R..		
N..	G2/G3	I..	J..	R..

5.33.5.4

A connecting circle between two concentric circles

Two concentric circles are a very special case of one circle inside the other one. In this case the centre points of both circles coincide. The word B1=.. which indicate the angle with the main axis of the line through the centre point of the concentric circles and the connecting circle, is used as additional information and has to be inserted in the block with the connecting circle.

The formats are:

Radius of the connecting circle is known

Two concentric circles

N..	G2/G3	I..	J..	{R..}
N..	G2/G3	R..	B1=..	
N..	G2/G3	I..	J..	

Radius of the second circle is known

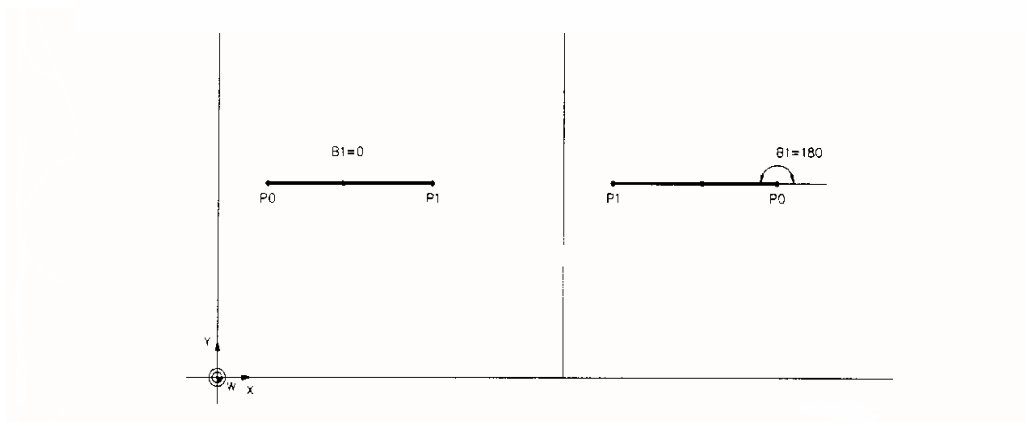
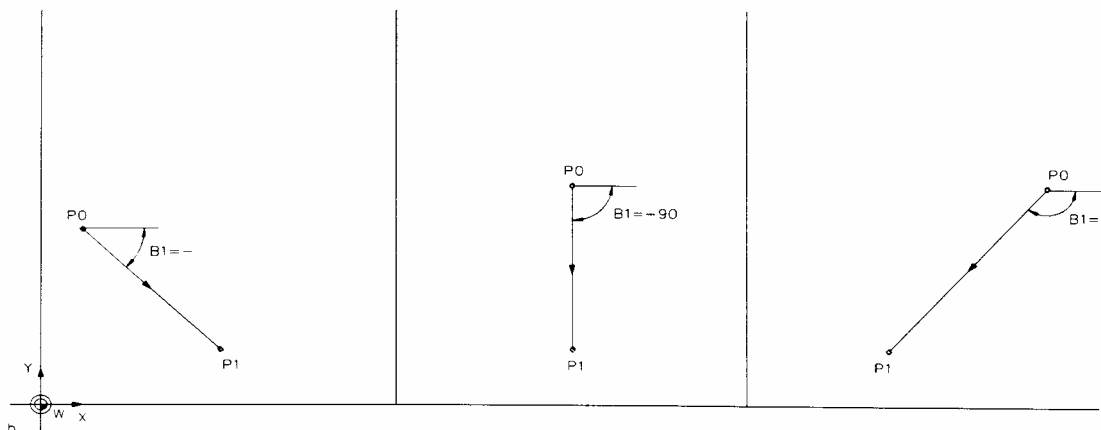
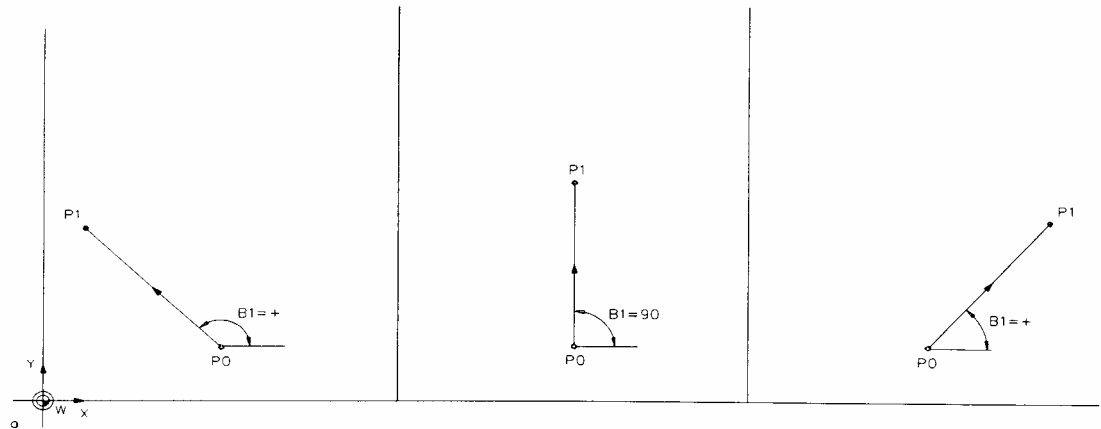
In this case the radius of the connecting circle will be calculated.

N..	G2/G3	I..	J..	{R..}
N..	G2/G3	B1=..		
N..	G2/G3	I..	J..	R..

5.33.6 Line definitions

Programming the angle b1=

In a lot of cases a linear movement has to be programmed with the angle, which the line makes with the main axis. The angle is programmed with the word B1=.. The angle should be programmed in the direction of movement, which means that one should look from the start point of the first movement to the end point of it. The sign of the angle contains the direction of movement and can be seen from the illustration.



Note: It is important that the angle is programmed correctly, otherwise the wrong intersection point can be chosen.

End point

An end point is programmed with:

- the absolute Cartesian coordinates X and Y
- the polar coordinates B2= and L2=
- a previously defined point P or P1=

In some cases the intersection point of two elements is known from the drawing and can be programmed as an end point. It is still possible to insert a chamfer between linear movements or a rounding between intersecting elements.

If the intersection point of two lines is programmed as an end point, this point is assumed to be the start point of the next movement and can be programmed with one (X.. or Y..) two coordinates (X.. and Y..) one coordinate and angle (X.. or Y.. and B1=..).

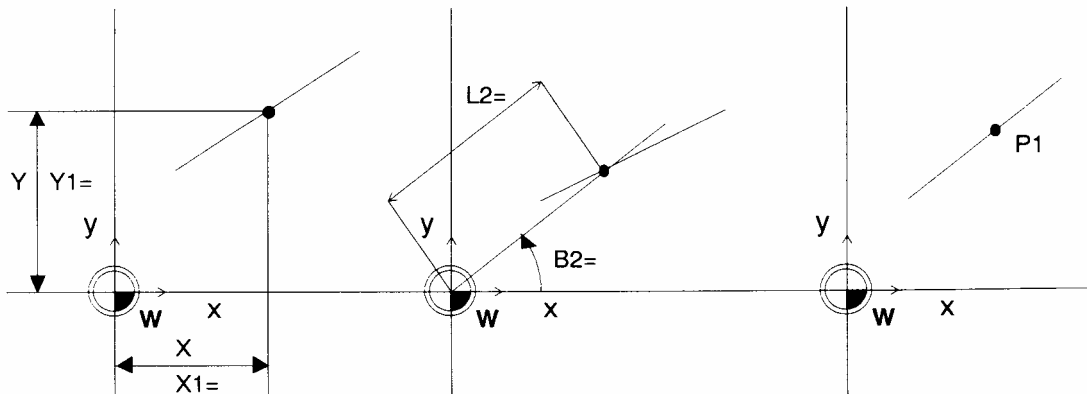
A horizontal or vertical line can also be programmed with one coordinate only. The other coordinate is picked up from the previous blocks. If the start point of the first element is not known, the angle which the line makes with the main axis, has to be added to the block.

Circle centre point

The circle centre is programmed with either the absolute Cartesian coordinates (I, J, K..) or its polar coordinates (B3=, L3=..).

Support point

When end point coordinates are unknown, another point on the same line can be used to support the calculations of the end point.



X.. Y.. I1=0 B2= L2= I1=0 P1 I1=0 X1=.. Y1=
Defining a support point.

Four formats are available:

N..	G1	{B1=..}	X1=..	Y1=..	
N..	G1	{B1=..}	X..	Y..	I1=0
N..	G1	{B1=..}	B2=..	L2=..	I1=0
N..	G1	{B1=..}	P..		I1=0

Any point on the line can be used as a support point. Its coordinates can be programmed with:

- the absolute Cartesian coordinates X1=, Y1= or X, Y
- the polar coordinates B2=, L2=
- a previously defined point P or P1=

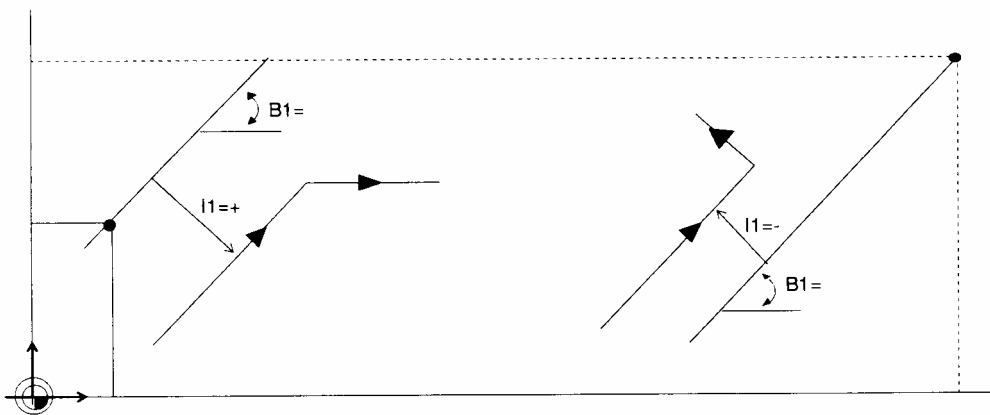
- Note:
1. If the end point coordinates can not be used due to the effect that the tool moves to the end point, a support point with I1=0 has to be programmed.
 2. If a support point is programmed and the line is not yet completely defined, the angle B1=.. which the line makes with the main axis, has to be programmed too. If a block contains too much information, an error message is displayed.

5.33.6.1 Parallel line

Sometimes a line is drawn parallel to a known line. The distance between the required line and the known one is programmed with the word I1=. The word I1= has a sign:

I1=+...: the line to the **right** of the existing line

I1=-...: the line to the **left** of the existing line



Defining a parallel line

The following formats are available:

N..	G1	B1=..	X..	Y..	I1=+/-..
N..	G1	B1=..	B2=..	L2=..	I1=+/-..
N..	G1	B1=..	P..	P1=..	I1=+/-..

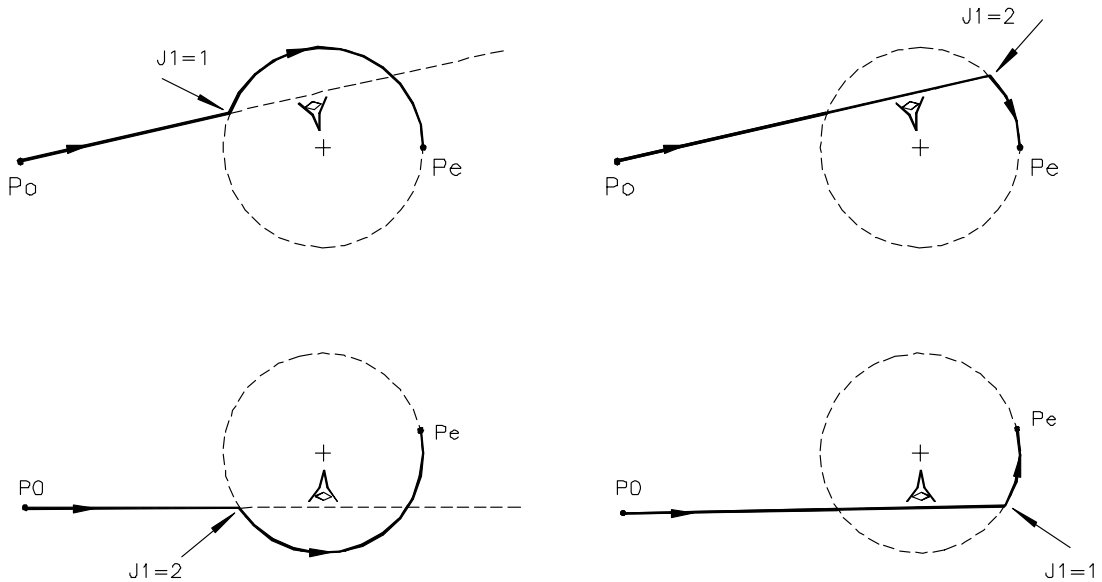
Any point on the existing line can be used. Its coordinates can be programmed with:

- the absolute Cartesian coordinates X, Y
- the polar coordinates B2=, L2=
- a previously defined point P or P1=

5.33.6.2 Intersection point indicator

When a line or circle or two circles cross each other, there will be two possible points of intersection. A special word (J1=1 or 2) is used to indicate which intersection point's coordinates must be calculated. Two main methods have to be used for determining which intersection point belongs to J1=1 and which one to J1=2.

1. when the line goes past the circle's centre

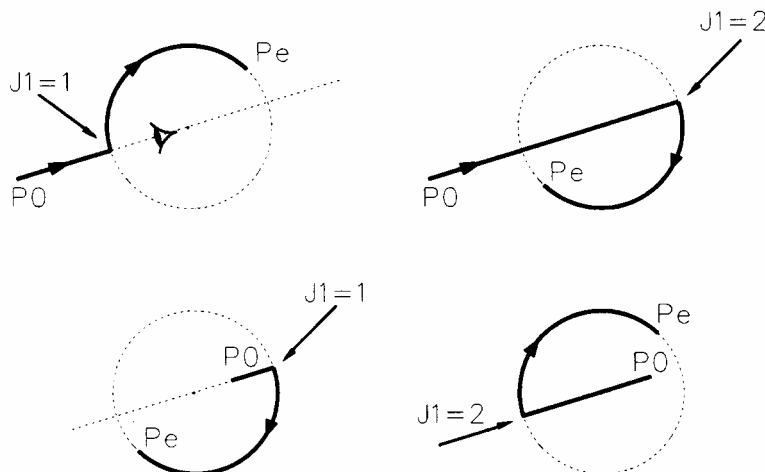


From the centre of the circle look at the line. The J1=1 intersection will be on the left and J1=2 intersection on the right of the perpendicular.

2. when the line goes through the centre of the circle

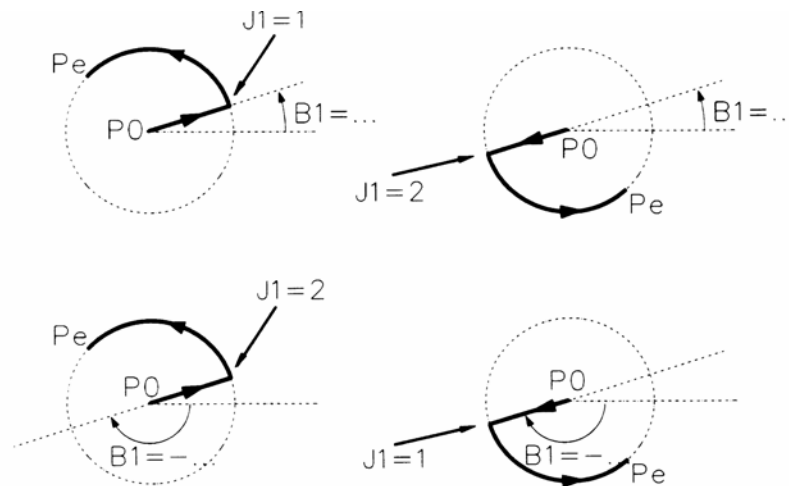
2.1 line intersects circle

a. start point of the line is not in the circle centre



The intersection point closest to the start point is J1=1; the other point is J1=2

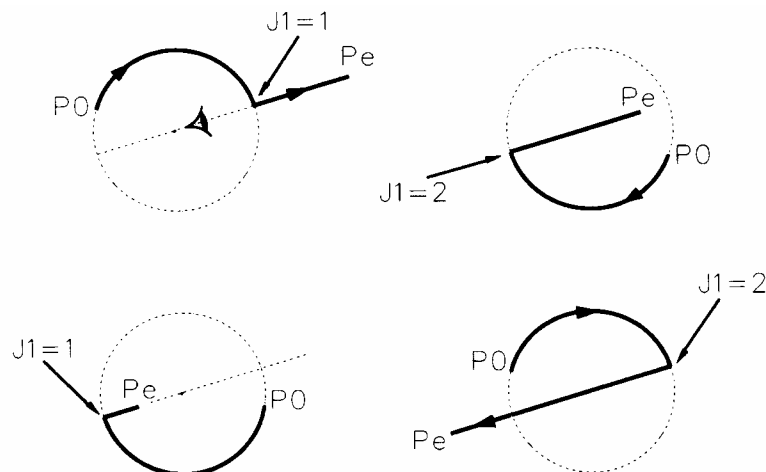
- b. start point of line is in the circle centre



The intersection point in the direction of movement on the line defines $J1=1$; the other point is $J1=2$.

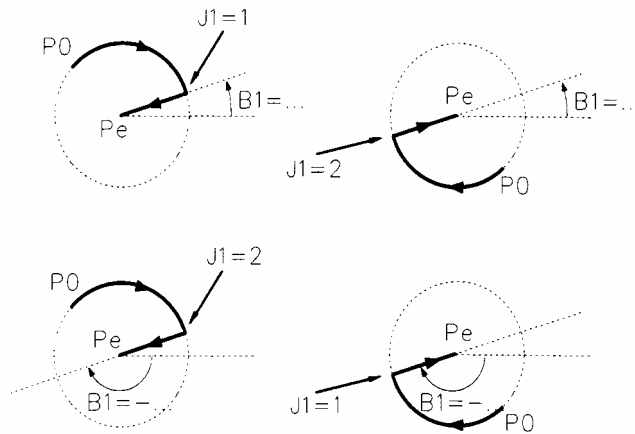
2.2 circle intersects line

- a. end point of the line is not in the circle centre



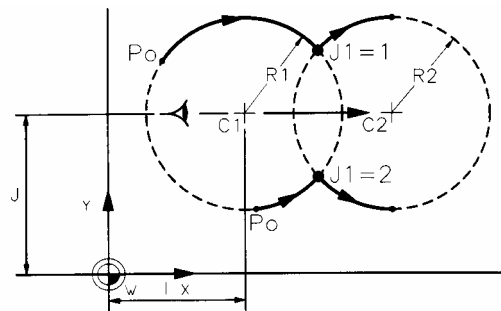
The intersection point closest to the end point is $J1=1$; the other point is $J1=2$

- b. end point of line is in the circle centre



The intersection point in the direction of movement on the circle defines $J1=1$; the other point is $J1=2$.

3. two circles intersect each other



When two circles intersect each other, the left ($J1=1$) or right ($J1=2$) intersection point is determined by looking from the centre point of the first circle to the centre point of the second one and seeing which intersection point is on the left or on the right from the line through the centres.

Note: Refer to PROGRAMMING THE ANGLE $B1=$ for the meaning of in the direction of movement on the line or circle.

5.33.7 Continuous and non-continuous movement

With a **continuous** movement the tool moves always in the forward direction. If more than one connecting circle is possible, it depends on the direction of movement on both elements, which connecting circle is automatically taken by the control as default circle. If a short or long arc with a connecting circle is possible, the shorter arc is chosen.

With a **non-continuous** movement:

- the tool can move backwards
- the toolpath can intersect itself
- the longer arc with a circular movement can be taken.

In some milling applications the non-continuous movements have to be used e.g. if the longer arc of a connecting circle should be programmed. In applications like laser cutting the non-continuous movements can be very useful.

Refer to chapter "Geometric calculations with continuous movements".

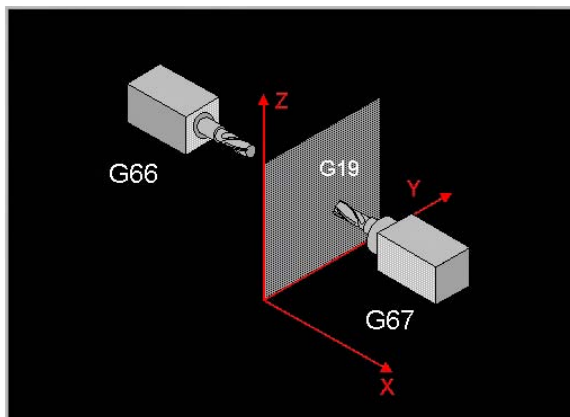
5.34 G66/G67 Select negative/positive tool direction

To select the direction in which the tool is pointing:
Tool length compensation in - direction/+ direction G66/G67.

G66: Tool is pointing in the negative direction of the tool axis

G67: Tool is pointing in the positive direction of the tool axis

The use of these functions allows a user always to enter a positive tool length value into the tool memory and the programmer to always look from the tool at the plane for circular interpolation and radius compensation



G T-length compensation in + dir.

Format

Tool pointing in the negative direction of the tool axis:

G66

Tool pointing in the positive direction of the tool axis:

G67

Notes and usage

Modality

G66 and G67 are modal functions.

Default mode

The function G66 is automatically activated when the CNC is switched on, thus for a tool pointing in the negative direction along the tool axis.

Availability

This G66/G67 function is not active in all versions, because it cannot be used on all machine types. It may also be that this function is only possible in G19.

Tool length in the tool memory

The tool length stored in the tool memory is always a positive value, unless corrections on the length of a standard tool are processed.

Tool length compensation

With G66 active (default mode) the tool length compensation is performed in the negative direction of the tool axis.

With G67 active the tool length compensation is performed in the positive direction of the tool axis.

Circular interpolation

To determine the direction of rotation on a circular the partprogrammer looks in the negative (G66) or positive (G67) direction of the tool axis at the plane in which the circle is made. In both cases G2 is used for a clockwise movement and G3 for a counter clockwise movement.

With G67 active the CNC makes the necessary conversions automatically during program execution.

Radius compensation

To determine if the tool is moving on the left or on the right of the workpiece the partprogrammer looks in the negative (G66) or positive (G67) direction of the tool axis at the plane in which radius compensation is made. In both cases G41 is used for the tool moving on the left and G42 for the tool moving on the right of the workpiece.

With G67 active the CNC makes the necessary conversions automatically during program execution.

Fixed cycles

If the tool is pointing in the positive direction of the tool axis, the depth of the fixed cycle must be programmed with a positive sign (+) to indicate that the cycle is to be executed in the positive direction of the tool axis. The sign is not automatically inverted.

With the milling cycles (G87 to G89):

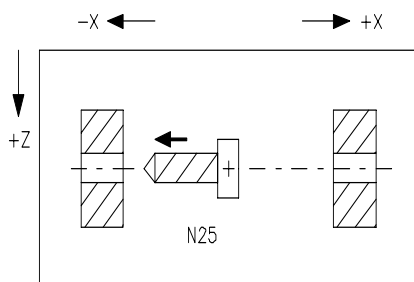
- The direction of rotation on the circular arcs is automatically changed in the opposite direction
- The milling direction programmed with the J-word, is not automatically changed.

Cancellation

Both functions cancel each other. CLEAR CONTROL, by M30 or by Softkey CANCEL PROGRAM, does not cancel them.

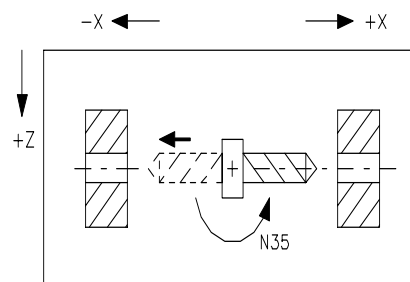
Restriction

Do not use the geometry (G64 active) in combination with G67. No error message is displayed, if the geometry is activated after G64. Contour errors may occur, because not all geometry functions are converted properly.

Example

G66 active

N25 G1 [End point coordinates]
N30 G67
N35 G1 [End point coordinates]



G67 active

First hole is drilled.
Select tool to point in the positive direction of the tool axis.
Second hole is drilled.

5.35 G70/G71 Inch/Metric programming

Allows the loading of partprograms, which use a different dimensional unit system from that, currently active in the CNC.

G70: dimensional units of the partprogram are in inches.

G71: dimensional units of the partprogram are in millimetres.

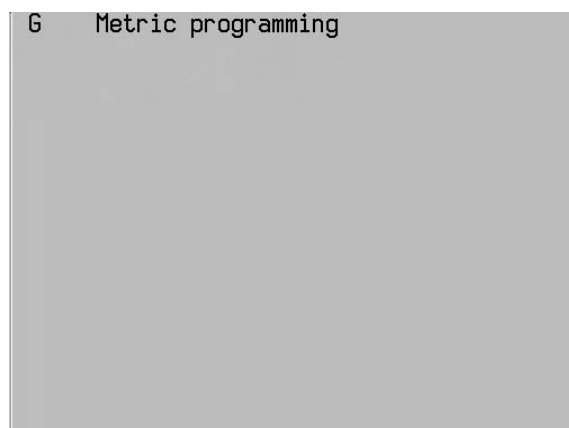
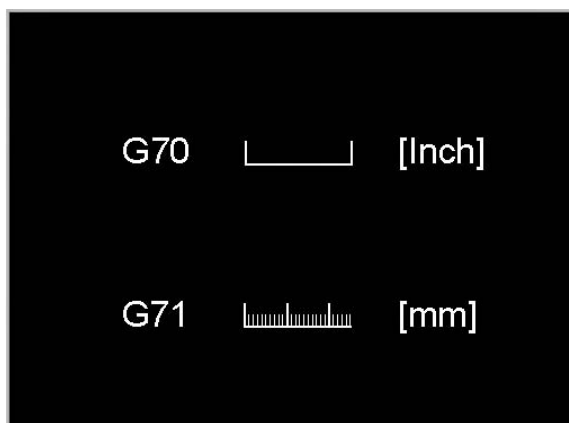
Format

Inch programming:

N... (PROG. NAME) G70

Metric programming:

N... (PROG. NAME) G71



Notes and usage

Modality

G66 and G67 are modal functions.

Dimensional units

Units for linear dimensions:	.001 mm	.0001 inch
Units for feedrate (G94):	.001 mm/min	.0001 inch/min
(G95):	.001 mm/rev	.0001 inch/rev
Units for cutting speed:	1 m/min	1 feet/min

Note: The cutting speed is used in the technology tables.

Active system of dimensional units

With a machine constant (MC707) is determined which type of dimensional units is used automatically by the CNC at initialising. This is the active system of dimensional units.

Note: The functions G70 and G71 are used at the input level of programmed data. They do not influence the measuring system on the machine tool.

Changing the system of dimensional units

If the active system of dimensional units has to be changed, e.g. if a partprogram in the other unit system should be entered via the keyboard, the machine constant setting has to be changed and the control reinitialised.

After initialising all dimensions in the memories are divided by 10 (changing from metric to inch) or multiplied by 10 (from inch to metric). Refer also to CNC MEMORIES.

Unit conversion at loading a program from a data carrier

If the CNC detects a G70 or G71 during the loading of a program from a data carrier, the CNC checks if the units used in the program and the active dimensional unit system are the same. If a difference is detected, the CNC converts the coordinates of the linear axes and the feedrates into the equivalents of the active system e.g. 'X1' (1 inch) is converted into 'X25.4' (25.4 mm). Also the function G70 or G71 is changed automatically by the control to the opposite function.

Note: Only one type of dimensional units is permitted in a program.

Executing a partprogram

If a G70 or G71 is not programmed at the beginning of a partprogram, the CNC assumes that all dimensions are in accordance with the unit system activated on the control.

If one of the functions G70 or G71 is programmed, the CNC checks if the stored program is in the same units as the active system of the control. If a difference occurs, an error is generated.

CNC memories

The CNC memories in which the tool dimensions, zero offsets, defined points and technology values are stored, must always be in the units of the active dimensional system. If this system is changed, all the stored values must be re-entered to their equivalents in the new unit system. The parameter memory is not influenced by a change to the other unit system.

Entering a program via the keyboard of the control

Programs, which are entered into the memory via the keyboard of the control, cannot contain a G70 or G71, which conflicts with the active dimensional system. If this is detected, an error message is generated.

Examples**Example 1**

CNC active system of unit's - metric
Partprogram values are in inches.

N9001 (EX.1) G70

N50 G1 X2 Y1.5 F8

Reading block N50 into the partprogram memory results in storing the coordinates X50.8 Y38.1 and a feedrate of 203.2 mm/min.

Example 2

CNC active system of unit's - inches.
Partprogram values are in millimetres.

N9002 G71

N50 G1 X50.8 Z38.1 F203.2

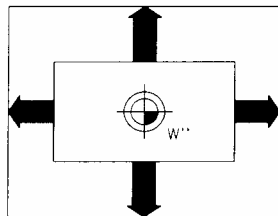
Reading block N50 into the partprogram memory results in storing the coordinates X2. Y1.5 and a feedrate of 8 inches/min.

5.36 G72/G73 Cancel/Activate scaling or mirror imaging

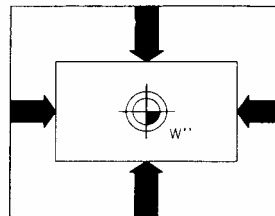
G72: To cancel scaling and mirror imaging.

G73: Activate scaling and/or mirror imaging.

- 1 To scale (enlarge or reduce in shape) a group of axis coordinates.

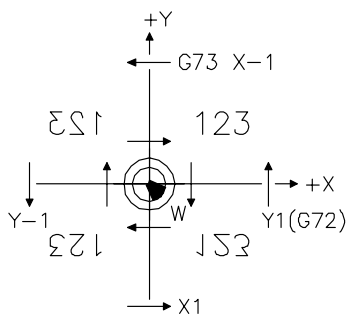


Enlargement

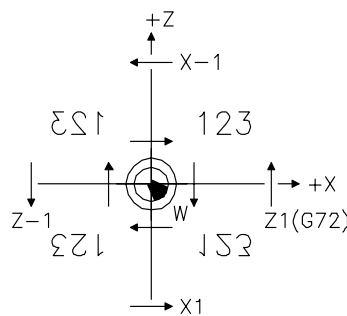


Reduction

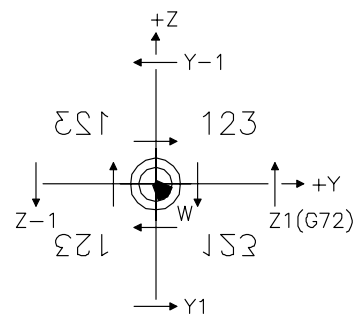
- 2 To produce a mirror image of a group of linear main axis coordinates or a change of sign of rotary axis coordinates. (sign inversion)



XY-plane (G17)



XZ-plane (G18)



YZ-plane (G19)

Format

To activate scaling

G73 A4=...

To cancel scaling

G73 A4=1 (factor) or A4=100 (percentage)

To produce a mirror image around an axis or a sign inversion of the axis.

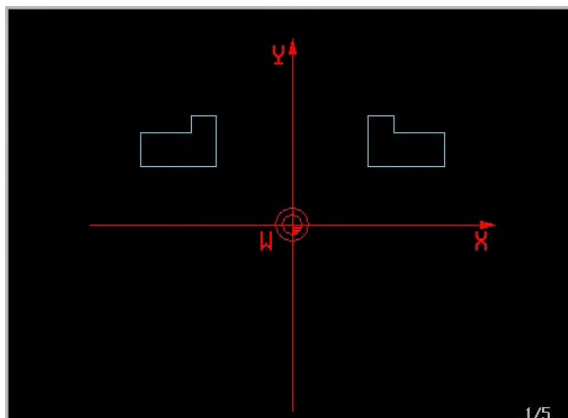
G73 {X-1} {Q-1} {Z-1} {A-1} {B-1} {C-1}

To cancel mirror image / sign inversion per axis

G73 {X1} {Y1} {Z1} {A1} {B1} {C1}

To cancel scaling and mirror image

G72



```
G Mirror image and scaling
X -1=set mirror image, 1=reset
Y -1=set mirror image, 1=reset
Z -1=set mirror image, 1=reset
B -1=set mirror image, 1=reset
C -1=set mirror image, 1=reset
A4= Scaling factor
```

Notes and usage**Modality**

G72 and G73 are modal functions.

Associated functions

G92/G93 axis rotation

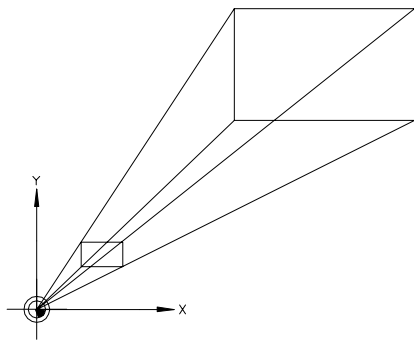
Scaling parameter A4=

The machine constant MC714 and MC715 determine if the A4= parameter is a factor (format 2.6) or a percentage (format 3.4).

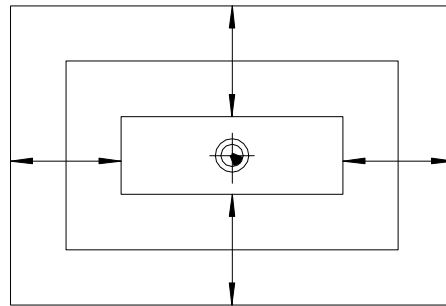
The format of the factor is set with another machine constant.

So a dimension increase of 1.25% is programmed as:

- A factor : G73 A4=1.0125
- A percentage : G73 A4=101.25

Geometric centre of the group of coordinates

Scaling about zero point W



Scaling about geometric centre

The scaling function uses the current zero point W as the starting point. If necessary, this point should be set by the use of a G92/G93 zero point shift at the geometric centre of the group of axis coordinates before the scaling operation. This ensures that the coordinates are symmetrically scaled around a fixed point, which is not moved out of position by the scaling operation.

Programmed zero point shifts (G92/G93)

G92/G93 zero point shifts are scaled if they are present in a group of coordinates to be scaled.

Programmed zero point shifts (G92/G93)

G92/G93 zero point shifts are scaled if they are present in a group of coordinates to be scaled.

G51-G59 zero point shifts

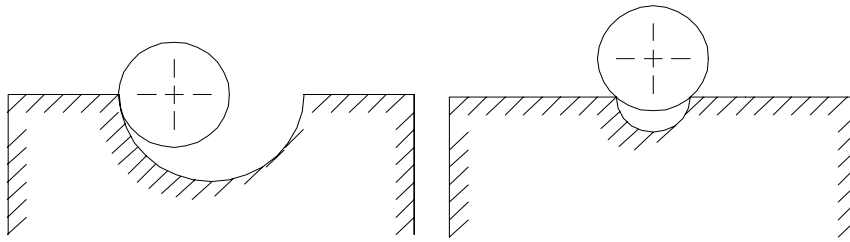
The G51-G59 zero point shifts are not influenced by the scaling operation.

Scaling the tool axis

With the machine constant for the factor is also determined if scaling is applied to only the axes coordinates in the main plane or to the tool axis as well.

Tool dimensions

If the tool axis is to be scaled, the tool length is not scaled. Tool diameters are not scaled.



Before scaling

After scaling - tool is too large

When scaling is to be performed, the programmer must decide if the existing tool diameter is suitable for the different dimensions.

Cancel scaling

The scaling is cancelled by:

- G72 mirror image if active, is cancelled too
- Softkey CLEAR CONTROL, M30 and Softkey CANCEL PROGRAM, both scaling and mirror image are cancelled
- G73 and the scaling factor: A4=1 or A4=100.

Mirror image**Sign inversion**

Mirroring around an axis is defined in the main-plane.

Mirroring around the Y-axis in the main-plane (G17 XY) means changing the sign of the X-coordinate in the opposite sign, thus +X to -X and vice versa (sign inversion).

The tool axis or rotary axis cannot be mirrored around an axis, but a sign inversion is still possible, thus +B to -B.

Plane selection

Mirroring has a meaning in the main plane only

G17 active:

- X-1: mirroring around Y-axis
- Y-1: mirroring around X-axis
- Z-1: sign inversion in tool axis

G18 active:

- X-1: mirroring around Z-axis
- Z-1: mirroring around X-axis
- Y-1: sign inversion in tool axis

G19 active:

- Y-1: mirroring around Z-axis
- Z-1: mirroring around Y-axis
- X-1: sign inversion in tool axis

Mirroring circular movements

When a circular movement is mirrored in one axis, its direction of rotation is also reversed: G2 becomes G3 and G3 becomes G2. This ensures that the tool travels in the correct direction when moving on circular arcs.

Tool radius compensation

Tool radius compensation is automatically reversed when mirroring occurs in one axis, for example: G41 becomes G42. This ensures that the tool radius compensation is correctly calculated from the programmed coordinates.

Programmed zero point shifts (G92/G93)

G92/G93 zero point shifts are mirrored too, if they are present in a group of coordinates to be mirrored.

G51-G59 zero point shifts

The stored G51-G59 zero point shifts are not influenced by the mirror operation.

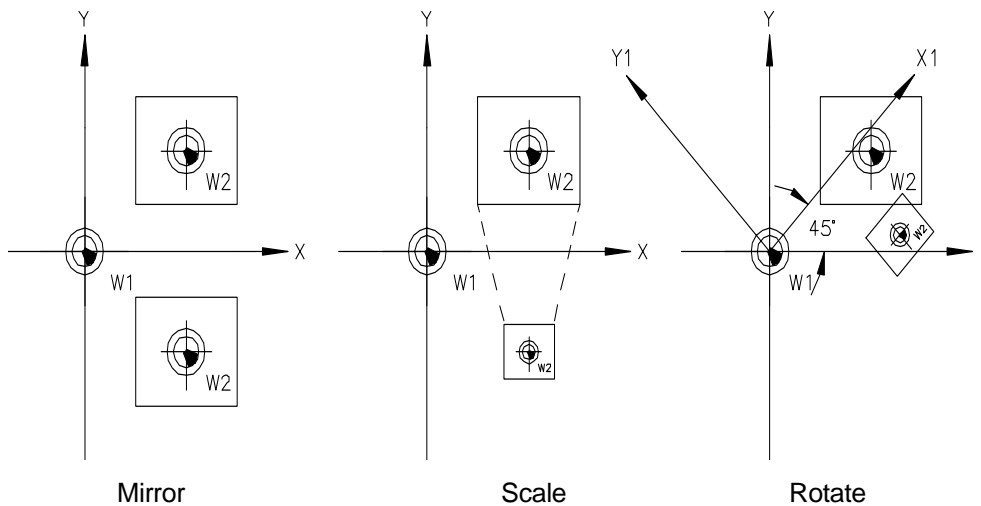
A G51-G59 zero point shifts are mirrored too, if they are programmed behind a G73.

Spindle rotation

The direction of spindle rotation is not reversed by the mirroring operation. The programmer must therefore consider this fact when deciding which axis coordinates are to be mirrored.

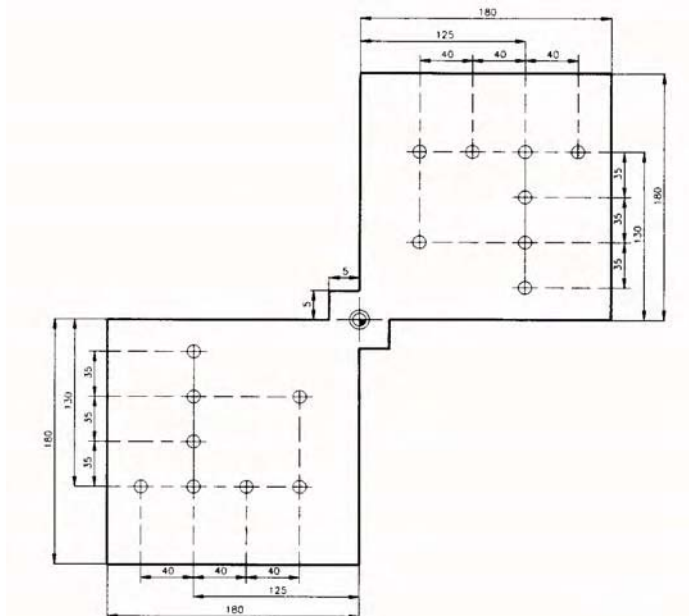
Scaling, mirror image and rotation of axes

A group of axes coordinates can be scaled, mirrored and rotated by using a combination of the G73 and G92/G93 functions with the word B4=.

**Cancel mirror image**

Mirror image is cancelled by:

- G72 a scaling operation, if active, is cancelled too
- Softkey CLEAR CONTROL, M30 and softkey CANCEL PROGRAM, both scaling and mirror image are cancelled
- G73 and the positive sign of the mirrored axis; E.g. X-1 is cancelled by X+1.

Example Mirror operation**N7273 (MIRROR IMAGE OF A POCKET)**

N1 G17

Select XY-plane.

N2 G54

Set program zero point.

N3 S300 T1 M6 (Mill radius 4 mm)

Load tool number 1.

N4 G0 X-5 Y10 Z10 F700 M3

Move tool rapidly to programmed position. Set feedrate to 700 mm/min and spindle rotation in clockwise direction.

N5 G1 Z-15

Feed tool to depth at set feedrate.

N6 G43 Y5

Move tool to programmed position.

N7 G41

Select tool radius compensation LEFT.

NS G1 X0

Machine the workpiece.

N9 G1 Y180

N10 G1 X180

N11 G1 Y0

N12 G1 X5

N13 G1 Y-10

N14 G40

Cancel tool radius compensation.

N15 G1 Z10

Retract tool from workpiece.

N16 G73 X-1 Y-1

Mirror coordinates around X- and Y-axis.

N17 G14 N1=4 N2=15

Repeat instructions from block 4 to 15.

N18 G72

Cancel the mirroring operation.

N19 S100 T2 M6 (Drill radius 4 mm)

Load tool number 2.

N20 G81 Y10 Z-20 F200 M3

Define drilling cycle and start spindle again.

N21 G79 Y60

Execute drilling cycle at programmed points.

N22 G79 Y95

N23 G79 Y130

N24 G79 X165

N25 G79 X85

N26 G79 X45

N27 G79 Y60

N28 G73 X-1 Y-1

Mirror coordinates around X- and Y-axis.

N29 G14 N1=21 N2=28

Repeat instructions from block 21 to 28.

N30 G72

Cancel the mirroring operation.

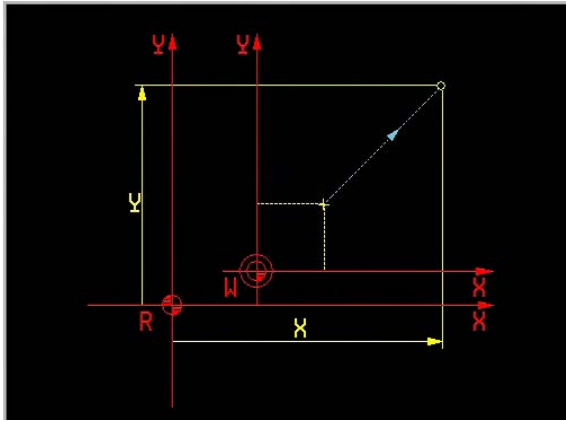
N31 G0 Z200 M30

Retract tool in tool axis and end of program.

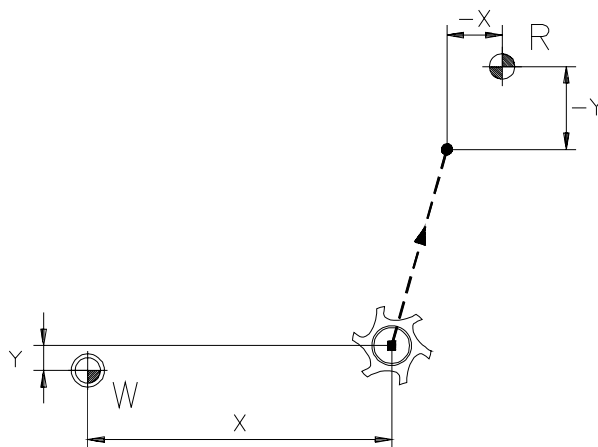
5.37 G74 Absolute position

To execute a rapid traverse movement to a position programmed with coordinates measured from the machine reference point R or machine positions.

G74 {X.. or X1=..} and/or {Y.. or Y1=..} and/or {Z.. or Z1=..} {K..} {L..} {K2=..}



G Absolute position
 X Endpoint coordinate
 Y Endpoint coordinate
 Z Endpoint coordinate
 B Endpoint angle
 C Endpoint angle
 K 0=inpod / 1=inpos / 2=MC or K2=
 L 0:with toollength, 1=without
 X1= Absolute position MC (1-18)
 Y1= Absolute position MC (1-18)
 Z1= Absolute position MC (1-18)
 B1= Absolute position MC (1-18)
 C1= Absolute position MC (1-18)
 K2= Inpoc window (0: MC, 1-32767 μm)



Notes and usage

Application

The G74-function's main application is in programming cycles for tool changers, pallet stations etc., when it is advisable that the programmed coordinates are independent of those used to define the machining of the workpiece.

Note: The absolute position is entered with the addresses X1=..., Y1=..., K2=.. etc for installation purposes!

End point coordinates

The end point coordinates can be defined in three different ways:

- 1) X100: relative position to the reference point.
- 2) For the first axis the machine positions 1 to 9 and 10 to 18 are describes in machine constants MC3145 – MC3154 and MC3158 – MC3165. For the second axis MC3245 -- MC3254 and MC3258 – MC3265. etc.
 When the actual machine constant is zero, no movement will be done.
- 3) X100 X1=...: relative position to Home position 2 (MC3146).

Stop between blocks (K-word)

All programmed axes move simultaneously during the execution of the G74. The next movement starts once all axes have reached their position. There is a stop between the G74-block and the next one as is usual with rapid traverse movements. (K0 is the default setting). With K1 the stop between the blocks can be avoided.

K0: Allowance is made for a (precise) stop between the movement of the G74 block and the movement in the next block, as is usual for rapid traverse movements. (K0 is the start setting).

K1: No allowance is made for a stop between the movement of the G74 block and the movement in the next block (corner rounding). The next movement is started when the desired position is nearly reached in all axes.

K2: No allowance is made for a stop between the movement of the G74 block and the movement in the next block. The next movement is started when the desired position is nearly reached in all axes. This position is defined by machine constant (MC136) (K2=0) or by the window size (K2=...).

K2= Display dimensions in mm (0-32.766 mm)

Incremental movements

If an incremental movement is programmed after a G74 movement, the coordinates are measured from the position stated in the G74-block.

Tool length compensation

In general tool length compensation is not used with the G74 positioning (L0 is default setting). If length compensation is required, 'L1' must be programmed.

Radius compensation

Radius compensation (G41 - G44) must be cancelled before the G74-function is activated.

Geometry function

The G64 geometry function must not be active when G74 is used.

Zero point shifts and zero offsets

The active zero point shift and zero offset are temporarily overridden.

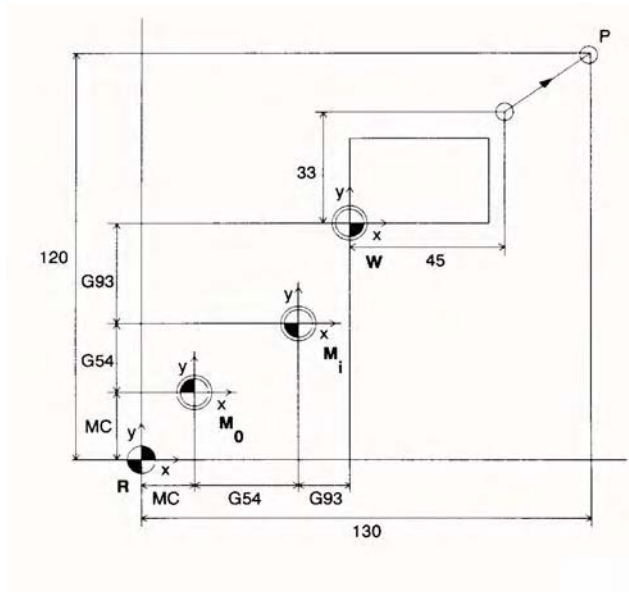
Axis rotation and scaling

The programmed G74-position is not affected by axis rotation or scaling.

After execution a G74 block

All zero points and the tool length compensation (if suppressed) become active again after a G74-block.

The last movement before G74 is activated must use either the G0 or G1 function. This function is automatically used in the first movement following the G74-block.

Example

From point P the coordinates with regard to R are known. The positioning to P is programmed as:

N10 G0 X45 Y33

N11 G74 X130 Y120

N20 G74 X100 X1=1 Y123.456 Z1=10 K2 K2=25.2

X100 X1=1

Y123.456

Z1=10 (Z0)

K2

Relative position to Home position from machine constants (MC3145).

Relative position to the reference point.

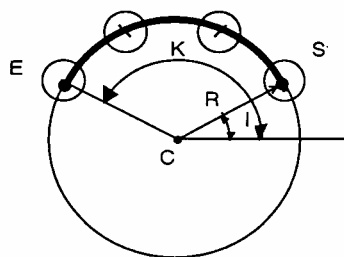
Position relative to Home position from machine constants (MC3554).

No allowance is made for a stop between the movement of the G74 block and the movement in the next block. The next movement is started when the desired position is nearly reached in all axes. This position is defined by the window size (K2=...).

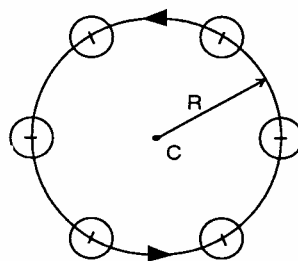
5.38 G77 Bolt hole cycle

To execute any fixed cycle (G81, G83-G89) at points which are equally spaced on a circular arc or a complete circle.

Format



circular arc
S = Startpoint
E = Endpoint



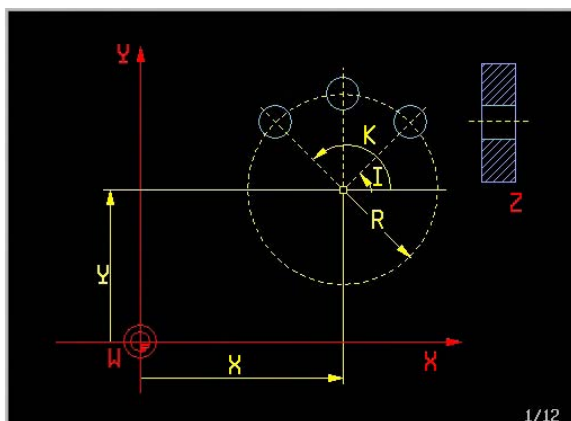
complete circle

Points on an arc

G77 [centre point coordinates] R... J...{I..}. {K..}.

Points on a complete circle

G77 [centre point coordinates] R... J... {I...}



G Bolt hole circle
X Center point coordinate
Y Center point coordinate
Z Center point coordinate
B Endpoint angle
C Endpoint angle
I Angle to first point
J Number of points
K Angle to last point
R Circular pattern radius
B1= Angle
B2= Polar angle
?90= Centre point abs. (X,Y,Z..)
?91= Centre point incr. (X,Y,Z..)
L1= Path length

L2= Polar length
P1= Point definition nr.for centre

Notes and usage

Associated functions

G79, G81, G83-G89

I angle

Minimum angle: - 360 degrees, maximum angle: + 360 degrees.

K angle

Minimum angle: 0 degrees, maximum angle: + 360 degrees.

The angle is programmed in degrees and decimal parts of a degree in steps of .001 degrees.

Direction of executing the points

When I-K greater than zero, the holes are in the CW direction.

When I-K smaller than zero, the holes are in the CCW direction.

Rotated pocket or groove (B1=)

A previously defined pocket or groove (G87 or G88) can be rotated about an angle. The centre of rotation is the point used in the G77 block to program the location of the pocket or groove.

The angle is programmed with the word B1= in degrees and decimal parts thereof and ranges from -360° to 360°.

The angle is measured with the X-axis (G17 and G18) or the -Z-axis (G19).

Three possibilities are available:

1. B1= not programmed in the G77 block. In this case the sides of the pocket or groove are parallel to the main axes.
2. B1=0 in G77 block. In this case the axis of each pocket or groove is radial, thus lies in the direction of the radius from the centre of the circle to the point on the circle. Refer to example 3 for programming this case.
3. B1<>0 In G77 block. In this case B1= indicates the angle which the pocket or groove makes with the radius to the centre of the pocket. Refer to example 4 for programming this case.

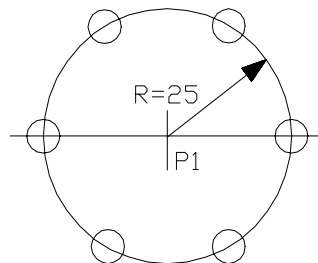
Note: The word B1= has two meanings in a G77 block. Either it is the angle for rotating a pocket or groove or it is used to program the coordinates (B1=, L1= or X/Y with B1=) for the position of the centre of the circle.

Kinematic calculations (G108)

When G108 is active, no rotary axes will be programmed (O141).

Examples

Example 1 Fixed cycle on a complete circle



N30 G78 P1 X... Y... Z...

N40 T1 M6

N50 G81 Y1 Z-10 F100 S1000 M3

N60 G77 P1 R25 I0 J6

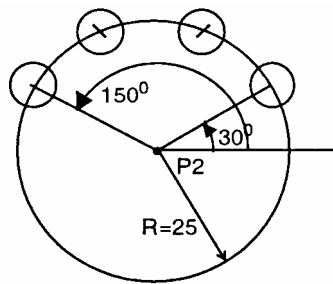
Definition of circle centre point (P1)

Load tool 1 (a drill)

Fixed cycle definition

Execute the fixed cycle on six points of the complete circle.

Example 2 Fixed cycle on an arc



N30 G78 P2 X... Y... Z...

N40 T1 M6

N50 G81 Y1 Z-10 F100 S1000 M3

N60 G77 P2 R25 I30 K150 J4

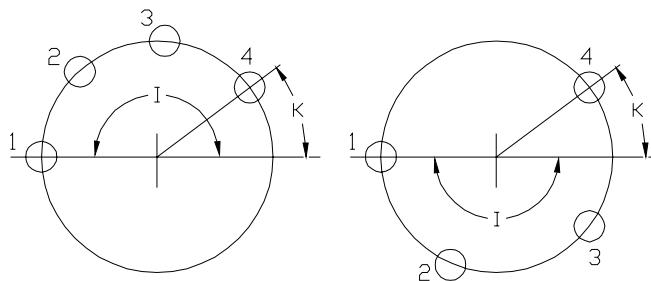
Definition of circle centre point (P2)

Load tool 1 (a drill)

Fixed cycle definition

Execute the fixed cycle on four equally spaced points on the circular arc, starting from 30

Example 3 Direction holes on an arc



I = 180
I-K > 0 CW

I = -180
I-K < 0 CCW

N50 G81 Y1 Z-10 F100 S1000 M3

N60 G77 X0 Y0 Z0 R25 I180 K30 J4

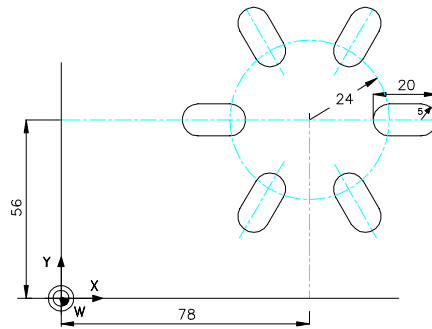
N70 G77 X0 Y0 Z0 R25 I-180 K30 J4

Cycle definition

Cycle repeating four times on an arc; Start on 180 degrees, End on 30 Grad in Clockwise (CW) direction.

Cycle repeating four times on an arc; Start on -180 degrees, End on 30 Grad in Counterclockwise (CCW) direction.

Example 4 Radial grooves



N60 T1 M6

Load tool 1, a mill with a radius of 4.8 mm

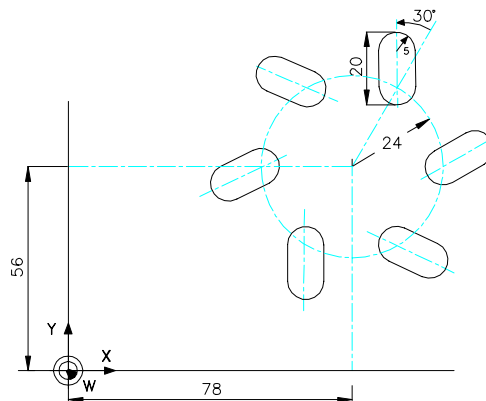
N65 G88 X20 Y10 Z-10 B1 F100 S1000 M3 Define the groove as if its sides are parallel to the X- and Y-axis

N70 G77 X78 Y56 Z0 R24 I0 J6 B1=0

The radial grooves are milled. This block contains:

- the centre of the bolthole circle (X78, Y56, Z0),
- the radius (R)
- the angle, which the radius of the first point makes with the X-axis (I)
- the number of holes on the circle (J)
- B1=0 to indicate that the grooves are radial.

Example 5 Rotated groove



N60 T1 M6

Load tool 1, a mill with a radius of 4.8 mm

N65 G88 X20 Y10 Z-10 B1 F100 S1000 M3 Define the groove as if its sides are parallel to the X- and Y-axis

N70 G77 X78 Y56 Z0 R24 I0 J6 B1=30

The rotated grooves are milled.

Refer to the previous example for an explanation of the addresses.

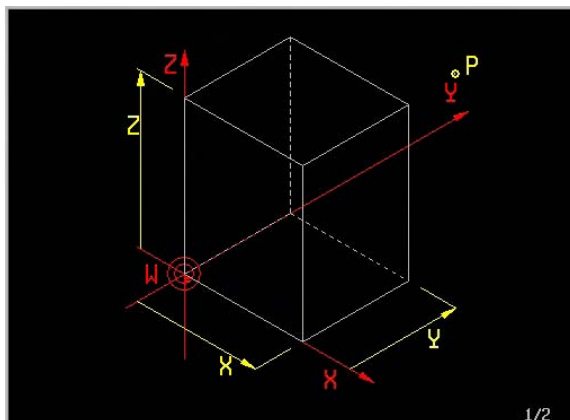
Only the word B1= has a different meaning.

5.39 G78 Point definition

Allows the coordinates of a point to be defined just once in a program. When a movement to the point is required, only the point number has to be programmed, not the point coordinates.

Format

G78 P... [Coordinates of point position]



G	Point definition
X	Point coordinate
Y	Point coordinate
Z	Point coordinate
B	Point angle
C	Point angle
P	Point definition number
B2=	Polar angle
L2=	Polar length

Notes and usage

Coordinates

Only Cartesian coordinates measured from the active program zero point W or polar coordinates (B2=, L2=..) in the main plane can be used.

G-functions with pre-defined points

The following G-functions can only contain one defined point in their program block: G2/G3, G77, G93.

The following G-functions can have a maximum of four defined points in their program block: G0, G1 and G79.

Using a pre-defined point

The format for using a predefined point is as follows:

N... G... P..., where P... represents the number of the point in the point memory.

Other formats are possible:

N... G79 P4=2 P2=10 P3=1 P1=5

N... G79 P1=E5 P2=E1

The P address can also be programmed with an index. The index value indicates the priority in the execution sequence. Index numbers 1 up to 4 are available (1=highest priority, 4=lowest priority). The number of the point in the point memory is entered behind the = sign.

Another option would be parameterised entry of the point definition. In this event the index also indicates the priority.

Point memory

A maximum of 255 defined points can be stored in the CNC's Point Memory; a Machine Constant sets this maximum.

Cancellation

A defined point's coordinates remain active until:

- the point is redefined again by another G78-block;
- the point memory is changed or cleared by the user;
- a data carrier with defined points is read in.

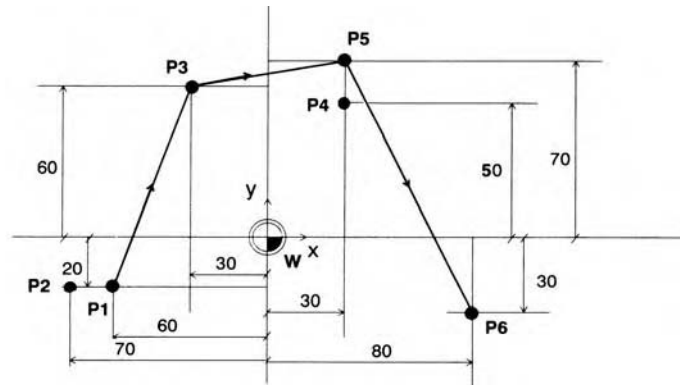
The Point Memory is not affected by CLEAR CONTROL.

Restrictions

Only one point can be specified in each G78-block, no other words are permitted.

Examples

Example 1.



```

N10 G78 P1 X=60 Y=20
N11 G78 P2 X=70 Y=20
N12 G78 P3 X=30 Y=60
N13 G78 P4 X=30 Y=50
N14 G78 P5 X=30 Y=70
N15 G78 P6 X=80 Y=30
:
N90 G0 P1=1
N91 G1 P1=3 P2=5 P3=6 F1000

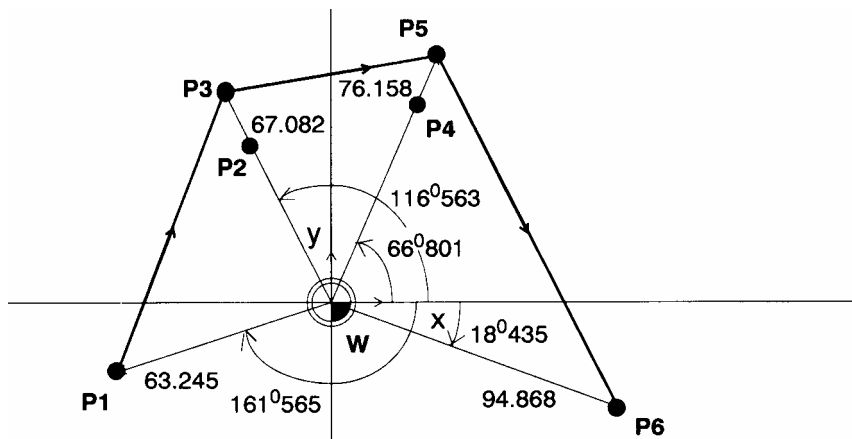
```

The points are defined.

Move tool rapidly to position defined by P1.

Move tool at set feedrate, first to P3, then to P5 and finally to P6.

Example 2 Using polar coordinates



```

N10 G78 P1 B2= -161.565 L2= 63.245
N11 G78 P2 B2= 116.563 L2= 65
N12 G78 P3 B2= 116.563 L2= 67.082
N13 G78 P4 B2= 66.801 L2= 72
N14 G78 P5 B2= 66.801 L2= 76.158
N15 G78 P6 B2= - 18.435 L2= 94.868
:
N90 G0 P1=1
N91 G1 P1=3 P2=5 P3=6 F1000

```

The points are defined.

Move tool rapidly to position defined by P1.

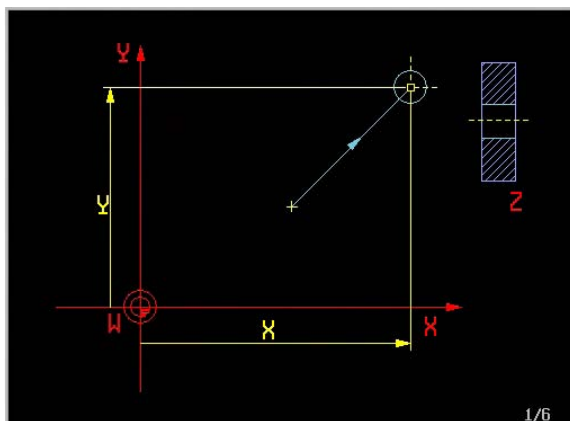
Move tool at set feedrate, first to P3, then to P5 and finally to P6.

5.40 G79 Activate cycle

To activate any fixed cycle which controls hole operations (G81, G83 to G86) or milling operations (G87 to G89) at programmed positions.

Format

N... G79 [Axis coordinates] {B1=...}



G Activate cycle
 X Point coordinate
 Y Point coordinate
 Z Point coordinate
 B Point angle
 C Point angle
 B1= Angle
 B2= Polar angle
 ?90= Point abs. (X,Y,Z..)
 ?91= Point incr. (X,Y,Z..)
 L1= Path length
 L2= Polar length
 P1= Point definition number
 P2= Point definition number
 P3= Point definition number

P4= Point definition number

Notes and usage

Associated functions

G77, G81, G83-G89

Executing defined fixed cycles

The positions where a previously defined fixed cycle is to be executed are programmed in the G79-blocks, which follow the fixed cycle definition.

A fixed cycle for a hole operation (G81, G83-G86) is executed in the tool axis which is perpendicular to the main plane defined by the G- function for plane selection (G17, G18 or G19).

A fixed milling cycle is executed in the main plane defined by the G- function for plane selection (G17, G18 or G19).

The first G79-block, which follows a defined fixed cycle, must contain a tool axis coordinate.

The direction of the depth operation is programmed in the fixed cycle block with the sign of the Z-word indicating the total depth.

Positioning logic

To reduce the possibility of collision between tool and workpiece the positioning logic is available. Refer to G0 POSITIONING LOGIC for additional information.

In addition, the programmer must ensure that the tool cannot collide with any device holding the workpiece in position.

Spindle rotation

If the spindle is not rotating when a G79-block is activated, an error message is generated and program execution stopped.

Radius compensation

When a G79-block is activated, radius compensation is cancelled by the CNC automatically generating the G40-function.

If radius compensation is required after a G79-block, the appropriate function (G41 -G44) must be programmed.

Rotated pocket or groove (B1=)

A previously defined pocket or groove (G87 or G88) can be rotated about an angle. The centre of rotation is the point used in the G79 block to program the location of the pocket or groove.

The angle is programmed with the word B1= in degrees and decimal parts thereof and ranges from - 360 to 360'.

The angle is measured with the X-axis (G17 and G18) or the -Z-axis (G19).

If B1=0 is programmed or B1= is not programmed at all, the pocket or groove are milled axis parallel.

Note: The word B1= has two meanings in a G79 block. Either it is the angle for rotating a pocket or groove or it is used to program the coordinates (B1=, L1= or X/Y, B1=) for the position of the centre of the pocket.

Group function (G0, G1, G2, G3) and G6

A G79-block ignores the functions G0, G1, G2, G3 and G6. When the G79-block execution is finished, the concerning function will become active again.

Tool pointing in positive direction (G66/G67)

If the tool is pointing in the positive direction of the tool axis (G67 activated), the depth of the fixed cycle must be programmed with a positive sign (+) to indicate that the cycle is to be executed in the positive direction of the tool axis.

With the milling cycles the direction of rotation on the circular arcs is automatically changed in the opposite direction.

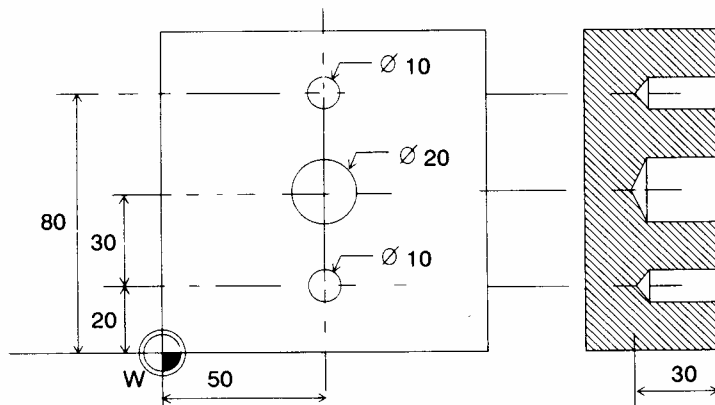
The milling direction programmed with the J-word, is not automatically changed.

Kinematic calculations (G108)

When G108 is active, no rotary axes will be programmed (O141).

Examples

Example 1 Three holes to be drilled



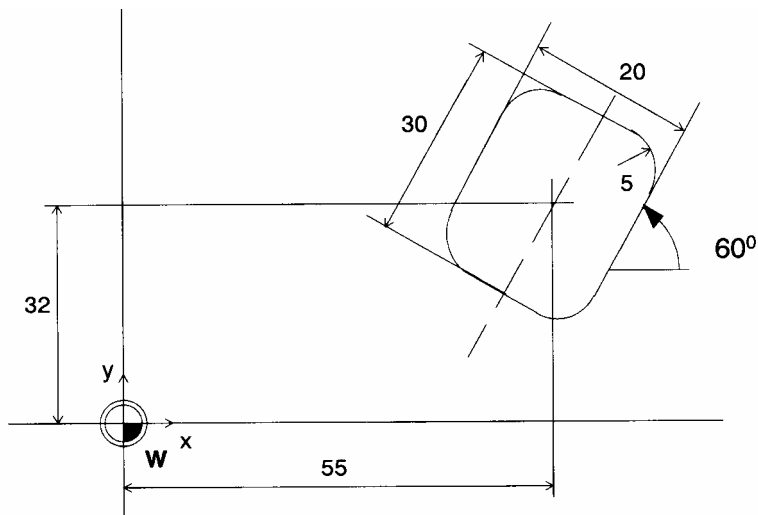
```

N50 G78 P1 X50 Y20 Z0
N55 G78 P2 X50 Y80 Z0
N60 T1 M6
N65 G81 Y1 Z-30 F100 S1000 M3
N70 G79 P1=1 P2=2
N75 T2 M6
N80 G79 X50 Y50 Z0
N90 M30

```

Define point 1
Define point 2
Load tool 1 (drill of diameter 10)
Define drilling cycle and start the spindle
Drill holes at points 1 and 2
Load tool 2 (drill of diameter 20)
Drill the hole
End of program.

Example 2 A rotated pocket



```

N55 G17
N60 T1 M6
N65 G87 X30 Y20 Z-5 B1 R5 F100 S1000 M3
N70 G79 X55 Y32 Z0 B1=60

```

Define the main plane for the pocket
Load tool 1, a mill with a radius of 4.5 mm
Define the pocket as if its sides are parallel to the X- and Y-axis
The pocket is milled. This block contains the centre of the pocket (X55, Y32, Z0) and the angle (60°) the axis of the pocket makes with the X-axis.

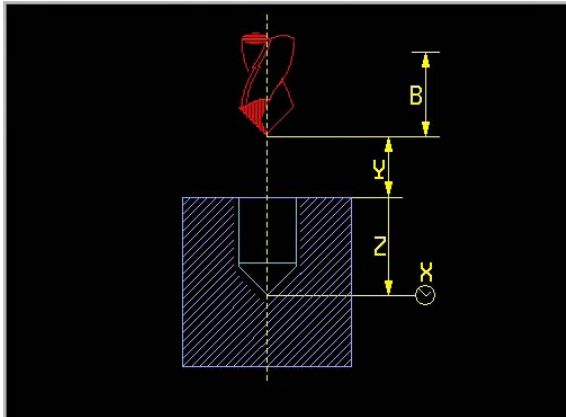
5.41 G81 Drilling cycle

To define in one program block the drilling of a hole.

Refer to the innovated cycle G781.

Format

G81 Z... {Y...} {X...} {B...}



G	Drilling cycle
X	Dwell time (sec)
Y	Clearance
Z	Drilling depth
B	Retract distance

Notes and usage

Associated functions

G77, G79, G83-G89

Depth of operation (Z)

Final depth of operation measured from the surface.

The sign of the Z-word indicates the direction of depth movement in the tool axis:

"-" in the negative direction, in most cases into the hole

"+" in the positive direction.

Retract distance (B)

The retract distance is added to the clearance value (Y-word). It can be used e.g. to avoid obstacles. This extra distance can have either a positive or negative value.

If the B-word is not programmed, the retract movement is executed to a point the clearance distance above the surface.

Dwell at bottom of hole (X)

If required, a dwell at the bottom of the hole can be programmed in steps of .1 second.

Minimum programmable dwell period: .1 second

Maximum programmable dwell period: 900 seconds

If the X-word is not programmed, no dwell is executed.

Execution

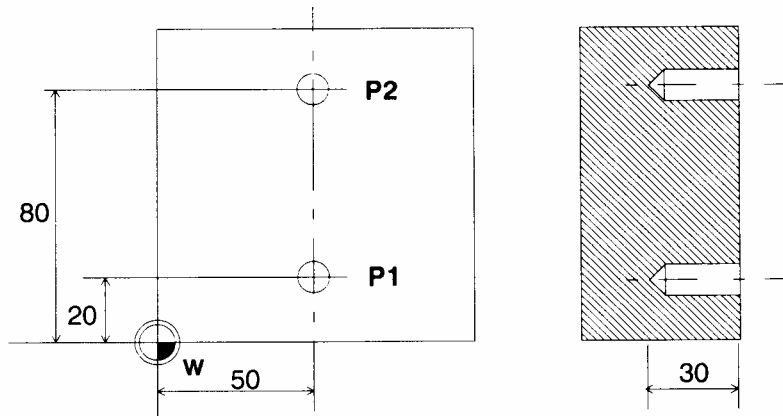
A defined fixed cycle is executed on the position(s) programmed in either the G77 or G79 block.

The cycle is executed in the tool axis, which is stated with the function for plane selection (G17, G18 or G19).

Cancellation

The cycle's values are cancelled when a new cycle is defined or by Softkey CLEAR CONTROL, M30 or by Softkey CANCEL PROGRAM.

Example



N50 G78 P1 X50 Y20 Z0

Define point 1

N55 G78 P2 X50 Y80 Z0

Define point 2

N60 G0 Z10 T1 M6

Load tool 1 and move tool out the tool change position

N65 G81 X1.5 Y1 Z-30 F100 S500 M3

Define fixed drilling cycle and start the spindle

N70 G79 P1 P2

Execute fixed cycle at point 1 and then point 2.

Fixed cycle sequence:

- drill moves at rapid traverse rate to a point the clearance distance (Y-word of G81) above the surface.
- drill feeds to depth (Z-word of G81) at set feedrate
- drill is stopped for 1.5 seconds
- drill is retracted at rapid traverse rate to a point the clearance distance above the surface.

5.42 G83 Deep hole drilling cycle

To define in one program block the drilling of a deep hole.

Refer to the innovated cycle G782 and G783.

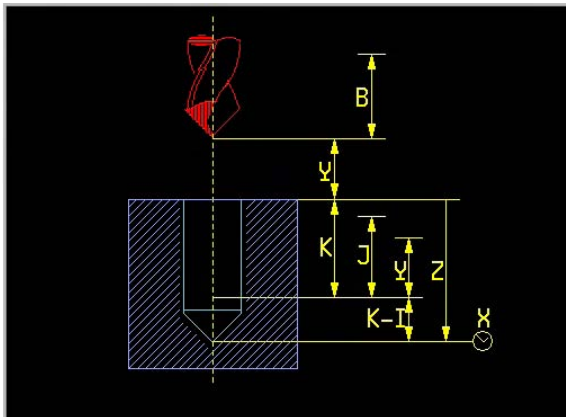
Format

Tool movements when J0 is active (tool is fully retracted to clearance position Y)

G83 Y... Z... K... J0 {I...} {X...} {B...}

Tool movements when J>0 (tool remains inside workpiece between cutting passes)

G83 Y... Z... K... J... {I...} {X...} {B...} {K1...}



G	Deep hole drilling cycle
X	Dwell time (sec)
Y	Clearance
Z	Overall drilling depth
B	Retract distance
I	Drilling depth decrement
J	Retract distance after step
K	Drilling depth first movement
K1=	Number of retract distances

Notes and usage

Associated functions

G77, G79, G81, G84-G89

Execution

A defined fixed cycle is executed on the position(s) programmed in either the G77 or G79 block.

The cycle is executed in the tool axis, which is stated with the function for plane selection (G17, G18 or G19).

Depth of operation (Z)

Final depth of operation measured from the surface. The sign of the Z-word indicates the direction of depth movement in the tool axis:

"-" in the negative direction, in most cases into the hole

"+" in the positive direction.

First feed-in distance (K)

In general a deep hole drilling operation takes place in several steps. The drilling depth of the first step is programmed with the K-word. If the K-value is greater than the total depth (Z-word), the hole is drilled at depth in one cutting pass.

The K-word has no sign.

Dwell at bottom of hole (X)

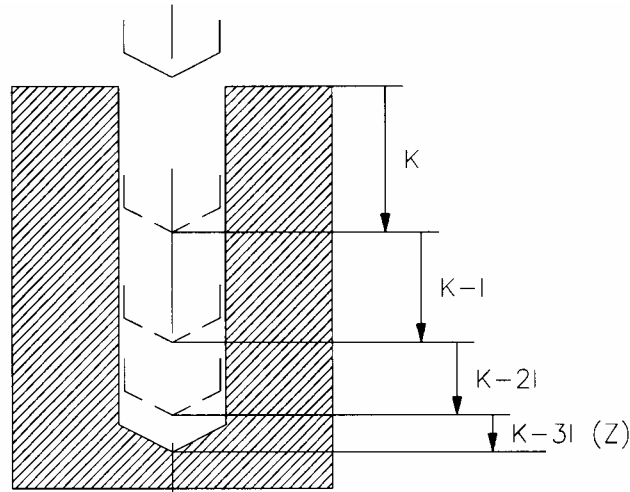
If required, a dwell at the bottom of the hole can be programmed in steps of .1 second.

Minimum programmable dwell period: .1 second

Maximum programmable dwell period: 900 seconds

If the X-word is not programmed, no dwell is executed.

Incremental reduction (I)



If the calculated feed-in distance becomes less than the I- value, the I- value is used instead. The final feed-in distance can be smaller than the I-value.

The I-word has no sign.

I0: all feed-in distances (except perhaps the final one) are equal to the first feed-in distance (K-word).

Retract distance (B)

The retract distance is added to the clearance value (Y-word). It can be used e.g. to avoid obstacles. This extra distance can have either a positive or negative value.

If the B-word is not programmed, the retract movement is executed to a point the clearance distance above the surface.

Retract indicator (J)

A special word (J) is used to indicate how the tool is to be retracted after each cutting pass:

J0: after each cutting step, the tool is retracted out of the hole to the point defined by the clearance distance.

J>0: after each cutting step, the tool is retracted over the programmed distance. In this way chips are broken, but the tool remains in the hole.

The J-word has no sign.

Number of special retract distance (K1=)

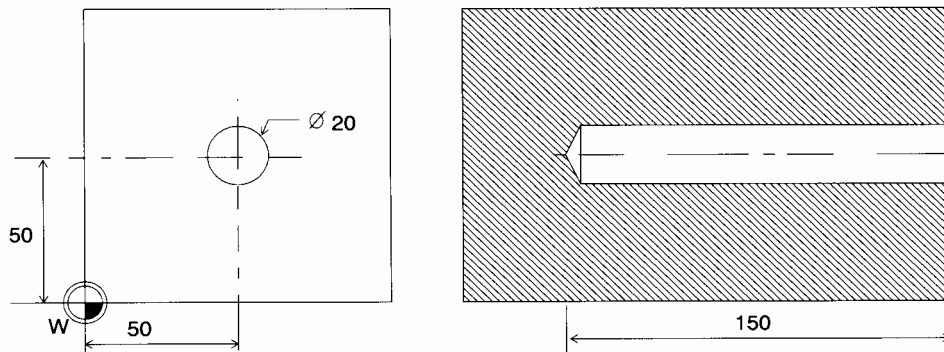
After a programmable number of feed movements defined by (K1 =), a retract will take place to a position the clearance distance before the preceding depth, to remove the chips. This only will take place if the special retract distance (J>0) is defined.

CANCELLATION

The cycles values are cancelled when a new cycle is defined or by Softkey CLEAR CONTROL, M30 or by Softkey CANCEL PROGRAM.

Examples

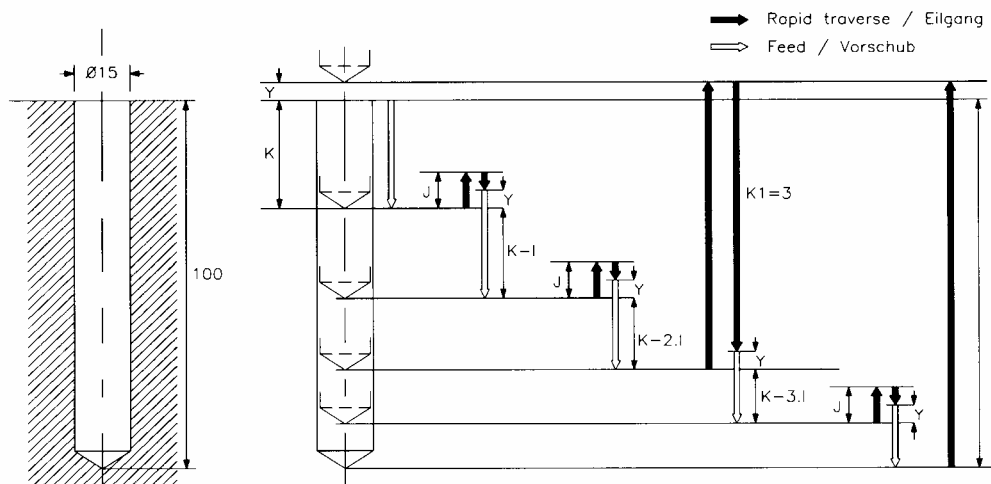
Example 1



N5 T1 M6
 N10 G83 Y4 Z-150 I2 J6 K20 F200 S500 M3
 N20 G79 X50 Y50 Z0

Load tool 1.
 Define the fixed cycle for deep hole drilling.
 Execute the fixed cycle at the programmed position

Example 2 Make a deep hole drilling ($d=15$, depth 100). After 3 feed movements a retract must take place.



N.. G83 Y3 Z-100 I5 J6 K30 K1=3

G83 Deep hole drilling cycle
 Y3 Clearance distance 3 mm
 Z-100 Depth of operation 100 mm
 I5 Incremental reduction 5 mm
 K30 First feed-in distance 30 mm. K-Value is reduced with the I-Value every cutting pass.
 K1=3 After 3 feed movements a retract will take place to the clearance position.

5.43 G84 Tapping cycle

Purpose

To define in one program block the tapping of a hole.

Refer to the innovated cycle G784 and G794.

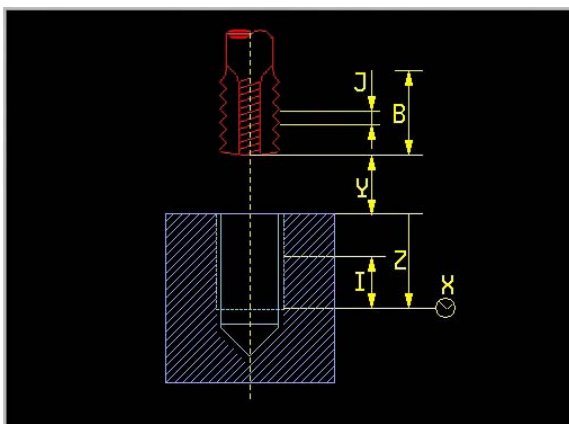
Format

G84 Z... {Y...} {B...} {J...} {X...}

or

G84 I1=0 Z... {Y...} {B...} {J...} {X...}

The tapping cycle can also be done as an interpolation in a close control circuit between the tool axis and the spindle. In this interpolation the acceleration power of the spindle is carrying out. There by is guaranteed, that the spindle is running with the desired speed. ("Rigid tapping").



G Tapping cycle
 X Dwell time (sec)
 Y Clearance
 Z Tapping depth
 B Retract distance
 I Speed ramp (rev)
 J Pitch
 I1= Interpolation (0=without,1=with)

I1= Interpolation (0 = without interpolation, 1 = with interpolation)

Notes and usage

Associated functions

G77, G79, G81, G83, G85-G89

Interpolation (I1=)

Tapping can be done without or with interpolation.

I1=0 without interpolation (default, open position control circuit)

I1=1 with interpolation (close position control circuit)

An active "tilting working plane (G7)" can work only with interpolation (I1=1).

It is also possible, by an active "tilting working plane (G7)" and when the tool head is not tilted (Tool axis is equal to Z-axis), to work without interpolation (I1=0).

Tapping depth (Z)

Final tapping depth measured from the surface. The sign of the Z-word indicates the direction of the movement in the tool axis:

Floating tap holder

When a floating tap holder is used, the clearance distance (Y) must be sufficient for the tool not to touch the workpiece when the tool is fully retracted and the tap holder spring is no longer under compression.

Retract distance (B)

The retract distance is added to the clearance value (Y-word). It can be used e.g. to avoid obstacles. This extra distance can have either a positive or negative value.
If the B-word is not programmed, the retract movement is executed to a point the clearance distance above the surface.

Rigid tapping

If a transducer is mounted at the spindle of the machine tool, rigid tapping, thus tapping without using a floating tapholder, is possible.

To eliminate the drift of the spindle an oriented spindle stop (M19) must be programmed before the tapping starts. On some machine tools the oriented spindle stop is automatically executed with a tool change (M6). Refer to the machine tool builder's documentation for details.

Retap thread

On machines with interpolation gives the programming of an oriented spindle stop (M19) the possibility to retap the thread.

Note After the interpolierende retap thread (I1=1) the modal M-function (M3, M4) is not active. This M function will be overwritten by M19.

Speed ramp (REV.) (I)

This word is used by the CNC to determine the point where the spindle starts to safely slow down before the end of the thread is reached and stop at the bottom of the thread.
With the I-address the number of revolutions required for the spindle to safely slow down and stop at the bottom of the thread is programmed. If the I-word is not programmed, the CNC uses a Machine Constant value (MC723) for establishing this point.

Pitch of the thread (J)

The pitch of the thread can be programmed:

- by using the J-word
- by programming the F (feedrate) = pitch (J) * spindle speed (S)

Dwell at bottom of hole (X)

If required, a dwell at the bottom of the hole can be programmed in steps of 0.1 seconds.

Minimum programmable dwell period: .1 second

Maximum programmable dwell period: 900 seconds

If the X-word is not programmed, the dwell time in MC724 is executed.

Minimum spindle speed

When feed and speed during G84 will be reducing to zero, it is possible that the spindle do not rotate. To avoid this is a minimum spindle speed in a machine constant (MC727) defined.

Execution

A defined fixed cycle is executed on the position(s) programmed in either the G77 or G79 block. The cycle is executed in the tool axis, which is stated with the function for plane selection (G17, G18 or G19).

Cancellation

The cycle's values are cancelled when a new cycle is defined or by Softkey CLEAR CONTROL, M30 or by Softkey CANCEL PROGRAM.

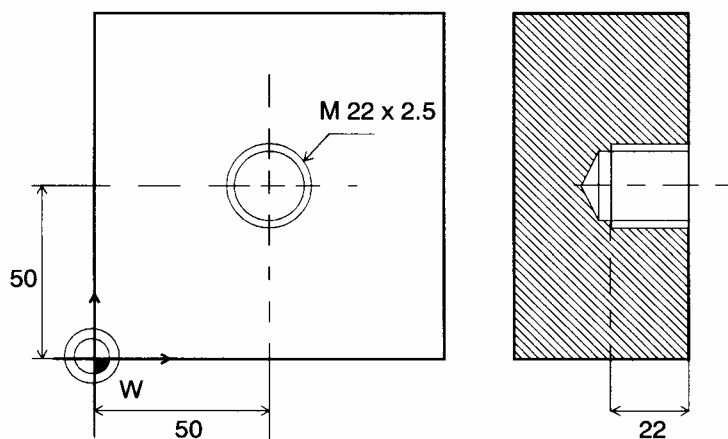
Note In case of G84 execution by G79 the control must be in G94 mode (F in mm/min) and not in G95 mode (F in mm/rev). The programmer should also always program G94 before G84.

Machine constant

The machine constant (MC723 and MC727) will not be use by interpolating.

The machine constant for the spindle must be defined properly. The acceleration of the spindle is calculated by machine constant MC2491, 2521,2551,2581 and MC2495, 2525, 2555, 2585. For an optimal control must be active also MC4430.

Example



N14 T3 M6

N15 G84 Y9 Z-22 J2.5 S56 M3 F140

N20 G79 X50 Y50 Z0

Load tool 3 (Tap M22 x 2.5).

Define tapping cycle and start the spindle.

Execute the tapping cycle on the programmed position. A floating tap holder is used.

Tapping cycle sequence:

- tap moves at rapid traverse rate to a point the clearance distance (Y-Word) above the surface. Spindle rotation clockwise (M3).
- tap feeds to tapping depth at a set feedrate. (The feed depends on pitch and spindle speed).
 - retract the tap to a point the clearance distance above the surface after the spindle rotation is switched to counter-clockwise.

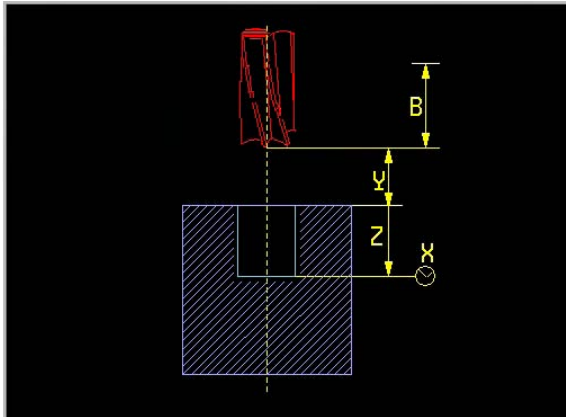
5.44 G85 Reaming cycle

To define in one program block the reaming of a hole.

Refer to the innovated cycle G785.

Format

G85 Z... {Y...} {B...} {X...} {F2=...}



G Reaming cycle
 X Dwell time (sec)
 Y Clearance
 Z Reaming depth
 B Retract distance
 F2= Feedrate to startpoint

Notes and usage

Associated functions

G77, G79, G81, G83, G84, G86-G89

Reaming depth (Z)

Reaming depth measured from the surface.

The sign of the Z-word indicates the direction of depth movement in the tool axis:

"-" in the negative direction, in most cases into the hole

"+" in the positive direction.

Retract distance (B)

The retract distance is added to the clearance value (Y-word). It can be used e.g. to avoid obstacles. This extra distance can have either a positive or negative value.

If the B-word is not programmed, the retract movement is executed to clearance

Feedrate to startpoint (F2=)

Feedrate to startpoint (F2=) is the feed from depth to starting position. This Feed (F2=) allows a faster retract feed and can be programmed. This will reduce the overall cycle execution duration. If the Feed F2= is not defined in a cycle the Feed to the starting position, is the programmed Feed (F).

Dwell at bottom of hole (X)

If required, a dwell at the bottom of the hole can be programmed in steps of .1 second.

Minimum programmable dwell period: .1 second

Maximum programmable dwell period: 900 seconds

If the X-word is not programmed, no dwell is executed.

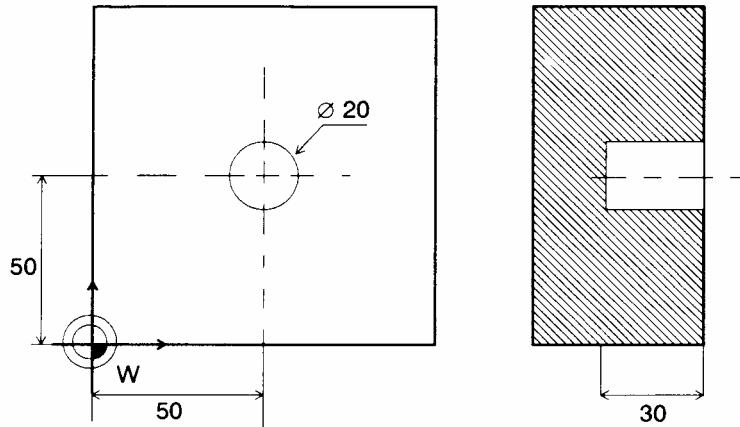
Execution

A defined fixed cycle is executed on the position(s) programmed in either the G77 or G79 block. The cycle is executed in the tool axis, which is stated with the function for plane selection (G17, G18 or G19).

Cancellation

The cycles values are cancelled when a new cycle is defined or by Softkey CLEAR CONTROL, M30 or by Softkey CANCEL PROGRAM.

Example



N25 Z10 T4 M6

N30 G85 X2 Y3 Z-20 F100 S1000 F2=200 M3

N35 G79 X50 Y50 Z0

Load tool 4, the reamer

Define the reaming cycle and start the spindle

Execute the reaming cycle at the programmed position

Cycle sequence:

- the reamer moves at rapid traverse rate to the clearance distance (Y3)
- the reamer feeds to depth (Z-20) at feedrate (F50)
- a dwell of 2 seconds (X2) at the bottom of the hole
- the reamer is retracted at feedrate (F2=200) to the clearance distance (Y3)

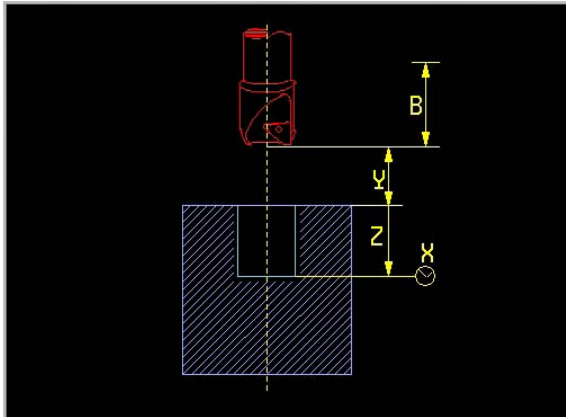
5.45 G86 Boring cycle

To define in one program block the boring of a hole.

Refer to the innovated cycle G786.

Format

G86 Z... {Y...} {X...} {B...}



G	Boring cycle
X	Dwell time (sec)
Y	Clearance
Z	Boring depth
B	Retract distance

Notes and usage

Associated functions

G77, G79, G81, G83-G85, G87-G89

Boring depth (Z)

Boring depth measured from the surface.

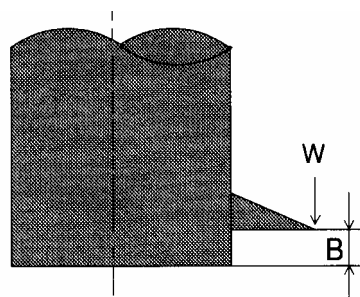
The sign of the Z-word indicates the direction of depth movement in the tool axis:

"-" in the negative direction, in most cases into the hole

"+" in the positive direction.

Retract distance (B)

The retract distance is added to the clearance value (Y-word). It can be used to compensate for a bore whose tool tip is not on the bottom of the boring bar.



W = tooltip

If this extra distance is not added, the bottom of the boring bar could still be inside the workpiece after the tool tip has been retracted the clearance distance.

If the B-word is not programmed, the tool tip is retracted to the clearance distance above the surface.

Dwell at bottom of hole (X)

If required, a dwell at the bottom of the hole can be programmed in steps of .1 second.

Minimum programmable dwell period: .1 second

Maximum programmable dwell period: 900 seconds

If the X-word is not programmed, no dwell is executed.

Execution

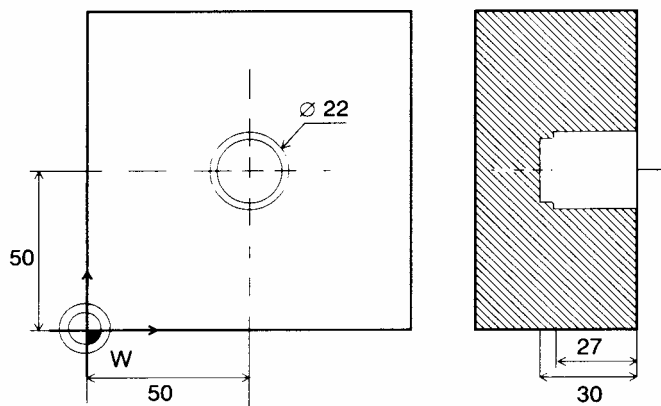
A defined fixed cycle is executed on the position(s) programmed in either the G77 or G79 block.

The cycle is executed in the tool axis, which is stated with the function for plane selection (G17, G18 or G19).

Cancellation

The cycles values are cancelled when a new cycle is defined or by softkey CLEAR CONTROL, M30 or by softkey CANCEL PROGRAM.

Example



N45 T5 M6

N50 G86 X1 Y9 Z-27 B10 F100 S500 M3

N55 G79 X50 Y50 Z0

Load tool 5, a boring bar.

Define boring cycle and start the spindle.

Execute fixed cycle at the programmed position

Fixed cycle sequence:

- the tool tip of the bore on the boring bar moves at rapid traverse rate to the clearance distance (Y-word).
- bore feeds to depth (Z-word) at feedrate
- at depth the spindle is stopped
- a dwell of 1 second
- the tool tip of the bore is retracted at rapid traverse rate to the clearance distance (Y-word).
- The tool tip is out of the hole.
- the spindle is started again
 - retract the tool tip to a point the retract distance (B-word) above the surface.

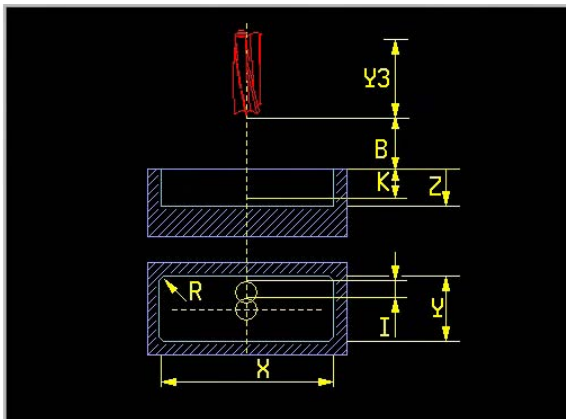
5.46 G87 Rectangular pocket milling

To define in one program block the geometry of a rectangular pocket and some parameters for cutting the pocket.

Refer to the innovated cycle G787 and G797.

Format

G87 X... Y... Z... {B...} {R...} {J...} {I...} {K...} {Y3=...} {F2=...}



G Rectangular pocket milling cycle
 X Dimension parallel to X
 Y Dimension parallel to Y
 Z Total pocket depth
 B Clearance
 I Cutting width mill in %
 J 1=climb, -1=conventional
 K Cutting depth
 R Corner radius
 Y3= Special retract distance
 F2= In depth feed (only this block)

Pocket geometry

X Dimension parallel to X
 Y Dimension parallel to Y
 Z Total pocket depth
 R Corner radius
 F2= In depth feed (only this block)

Cutting parameters

B Clearance
 I Cutting width mill in %
 J J1: climbing / J-1: conventional
 K Cutting depth
 Y3= Special retract distance

Notes and usage

Associated Functions

G77, G79, G81, G83-G86, G88, G89

Total pocket depth (Z)

Total pocket depth measured from the surface.

The sign of the Z-word indicates the direction of depth movement in the tool axis:

"-" in the negative direction, in most cases into the hole

"+" in the positive direction.

Plane of operation

The table below lists which axes are involved with the words X and Y to define the pocket dimensions in each of the three main planes.

	XY-PLANE	XZ-PLANE	YZ- PLANE
X-word parallel to	X-axis	X-axis	Z-axis
Y-word parallel to	Y-axis	Z-axis	Y-axis
Z-word (tool axis)	Z-axis	Y-axis	X-axis

The X and Y-word are programmed without sign.

Feedrate to cutting depth (F2=)

Feedrate to cutting depth (F2=) is the feed of the depth movement in the toolaxis for each cleaning pass.

If the Feed F2= is not defined in a cycle the Feed to the depth (K-word), is half the programmed feed (F).

Cutting width in % (I)

The value of the I-word states the percentage of the tool diameter to be used as cutting width for each cutting pass.

For example: I75 states that the cutting width is equal to 75% of the tool diameter.

If the I-word is not programmed, the value of a machine constant (MC720) is used.

Cutting depth (K)

If the pocket can not be cleaned out at depth in one pass, the K-word is used to program the depth for each cleaning pass.

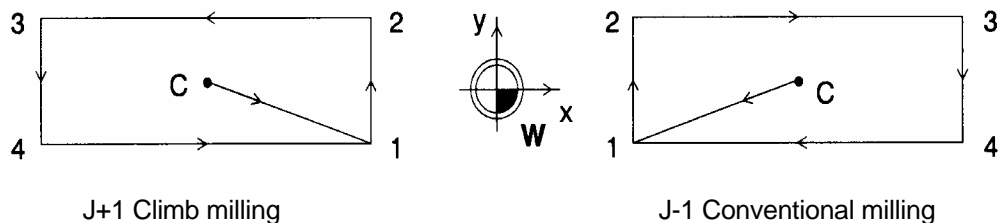
If the K-word is not programmed or has the same value as the Z-word, the groove is milled in one pass.

Special retract distance (Y3=)

The special retract distance Y3= is added to the clearance distance (B-word) this gives the opportunity to avoid objects. The special retract distance can have a sign, which will normally be the positive one.

If the special retract distance (Y3=) is not defined a retract movement is executed to the clearance distance.

Direction of milling (J)



If no direction of milling is programmed, climb milling is assumed (J1 = default direction).

Execution

A defined fixed cycle is executed on the position(s) programmed in either the G77 or G79 block. In these blocks the coordinates of the entry point of the pocket are stated.

The pocket is milled in the active plane (G17, G18 or G19). The radius of the actual tool stored in the tool memory, is used for milling the pocket.

Cycles must be programmed in G40 mode.

Rotated pocket

If the pocket is to be milled making an angle with the main axes, the G77 and G79 block are extended with a special word (B1=) to indicate the angle of rotation. Refer to the functions G77 and G79 ROTATED POCKET OR GROOVE for additional information.

Note: In the cycle the pocket is defined as if it is parallel to the axes.

Cancellation

The cycle's values are cancelled when a new cycle is defined or by softkey CLEAR CONTROL, M30 or by CANCEL PROGRAM.

Finishing the pocket

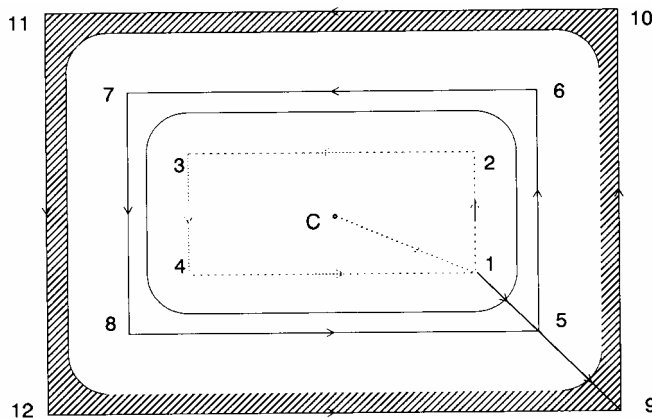
If a pocket must be milled and must have a finishing pass the following method may be used:

1. Add the allowance of the finishing cut to the tool radius and store the larger radius in the tool memory.
2. Execute the G87 milling cycle. A contour is milled which is smaller than the programmed contour due to the difference between the tool memory radius and the actual radius.
3. Program the contour of the pocket with the regular G1 and G2/G3 functions and execute the finishing pass with radius compensation and use the actual tool diameter.

The tool sequence

The tool sequence for milling the pocket is:

- a. With rapid traverse to the centre (C) of the pocket and stay the clearance distance (B-word) above the workpiece.
- b. With half the programmed feed to the first depth (K-word).



Pocket milling sequence

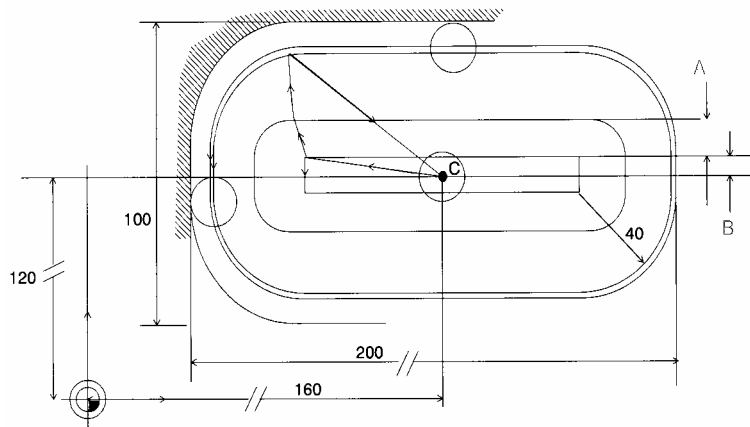
- c. Move the tool from the centre to point 1 and mill around from 1 to 2, 3, 4 and back to 1. Point 1 is calculated by the control and depends on the X-word, Y-word and the radius of the active tool.
- d. Move the tool to point 5. The point is calculated by the control. The distances parallel to the axes are: I-word x tool diameter
- e. Move the tool around from 5 to 6, 7, 8 and back to 5.
- f. Repeat the steps d and e -if necessary- until the layer is cleaned out.
- g. Finally follow the programmed contour and stop in the centre of the corner.
- h. If the programmed depth is reached, retract the tool to the clearance.
- i. If the programmed depth is not reached, move, with three times the programmed feed, to the centre (C) of the pocket.
- j. Clean out another layer by repeating the steps b to i

After the cleaning out, a finishing for the sides of the pocket might be necessary.

The best way to proceed is to store in the tool memory, for the actual tool, a radius being the stock removal greater than the actual radius of the tool. Once the cycle is totally executed this stock removal remains for finishing. Activating the tool radius compensation and using the regular G1 and G2/G3 blocks program the finishing of the pocket.

Examples

Example 1

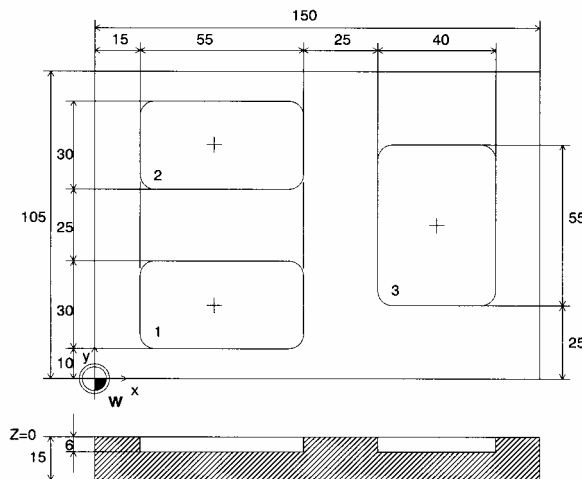


A: 75% of the diameter
B: 75% of the radius

N10 T1 M6 (Mill R5)
N20 G87 X200 Y100 Z-6 J+1 B1 R40
I75 K1.5 F200 S500 M3
N30 G79 X160 Y120 Z0

Load tool 1.
Define pocket milling cycle.
Execute cycle at programmed position.

Example 2 Example with three pockets.



N10 T1 M6 (Fräser R5)
N20 G87 X55 Y30 Z-6 J+1 B1 I75 K1.5 F200 S500 M3
N30 G79 X42.5 Y25 Z0
N31 G79 X42.5 Y80 Z0
N40 G87 X40 Y55 Z-6 J+1 B1 I75 K1.5 F200 S500 M3
N50 G79 X115 Y52.5 Z0

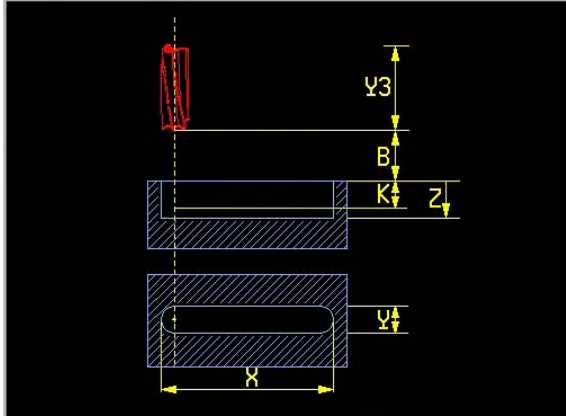
5.47 G88 Groove milling cycle

To define in one program block the geometry of a groove and some parameters for milling it.

Refer to the innovated cycle G788 and G798.

Format

G88 X... Y... Z... {B...} {J...} {K...} {Y3=..} {F2=..}



G Groove milling cycle
 X Dimension parallel to X
 Y Dimension parallel to Y
 Z Total groove depth
 B Clearance
 J 1=climb, -1=conventional
 K Cutting depth
 Y3= Special retract distance
 F2= In depth feed (only this block)

Groove geometry

X Dimension parallel to X
 Y Dimension parallel to Y
 Z Total groove depth
 F2= In depth feed (only this block)

Cutting parameters

B Clearance
 J J1: climbing / J-1: conventional
 K Cutting depth
 Y3= Special retract distance

Notes and usage

Associated functions

G77, G79, G81, G83-G87, G89

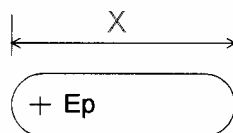
Total depth (Z)

Total depth of the groove measured from the surface.

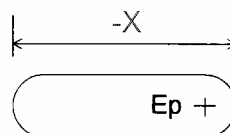
The sign of the Z-word indicates the direction of depth movement in the tool axis:

"-" in the negative direction, in most cases into the hole
 "+" in the positive direction.

Groove parallel to X-axis



X... (G17), X... (G18), Y... (G19)



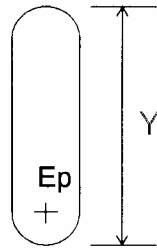
X-... (G17), X-... (G18), Y-... (G19)

If the axis of the groove should be parallel to the X-axis, the programming is as follows:

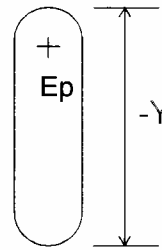
Absolute value of X-word > than value of Y-word

The sign of the X-word determines on which side of the entry point (Ep) the groove is milled. The Y-word is programmed without sign.

Groove parallel to Y-axis



Y... (G17), Y... (G18), X... (G19)



Y... (G17), Y... (G18), X... (G19)

If the axis of the groove should be parallel to the Y-axis, the programming is as follows: absolute value of-Y-word > than value of X-word

The sign of the Y-word determines on which side of the entry point (Ep) the groove is milled. The X-word is programmed without sign.

Plane of operation

The table below lists which axes are involved with the words X and Y to define the length and width of a groove in the three main planes.

	XY-PLANE	XZ-PLANE	YZ-PLANE
X-word parallel to	X-axis	X-axis	Z-axis
Y-word parallel to	Y-axis	Z-axis	Y-axis
Z-word (tool axis)	Z-axis	Y-axis	X-axis

Feedrate to cutting depth (F2=)

Feedrate to cutting depth (F2=) is the feed of the depth movement in the Toolaxis for each cleaning pass.

If the Feed F2= is not defined in a cycle the Feed to the depth (K-word), is half the programmed feed (F).

Cutting depth (K)

If the groove cannot be cleaned out at depth in one pass, the K-word is used to program the depth for each cleaning pass.

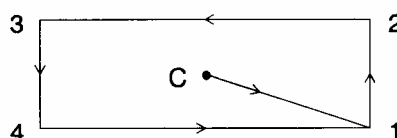
If the K-word is not programmed or has the same value as the Z-word, the groove is milled in one pass.

Special retract distance (Y3=)

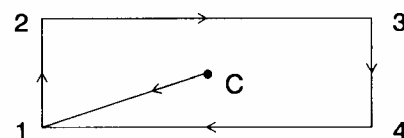
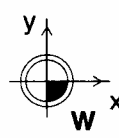
The special retract distance Y3= is added to the clearance distance (B-word) this gives the opportunity to avoid objects. The special retract distance can have a sign, which will normally be the positive one.

If the special retract distance (Y3=) is not defined a retract movement is executed to the clearance distance.

Direction of milling on the finishing path (J)



J+1 Climb milling



J-1 Conventional milling

If no direction of milling is programmed, climb milling is assumed (J1 = default direction).

Execution

A defined fixed cycle is executed on the position(s) programmed in either the G77 or G79 block. In these blocks the coordinates of the entry point of the groove are stated.

The groove is milled in the active plane (G17, G18 or G19). The radius of the actual tool stored in the tool memory, is used for milling the groove.
Cycles must be programmed in G40 mode.

Rotated groove

If the groove is to be milled making an angle with the main axes, the G77 and G79 block are extended with a special word (B1=) to indicate the angle of rotation. Refer to the functions G77 and G79 ROTATED POCKET OR GROOVE for additional information.

Note: In the cycle the groove is defined as if it is parallel to the axes.

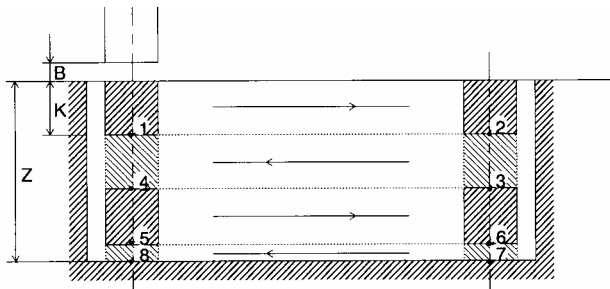
Cancellation

The cycle's values are cancelled when a new cycle is defined or by softkey CLEAR CONTROL, M30 or by CANCEL PROGRAM.

The tool sequence

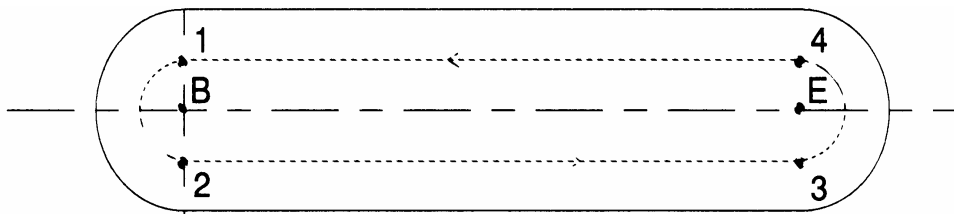
The tool sequence for milling the groove is:

- a. With rapid to point B where the tool enters the groove and stay the clearance distance (B-word) above the workpiece.
- b. With half the programmed feed to the first depth (1).
- c. With the programmed feed through the centre of the groove to point E (2).
- d. With half the programmed feed to the second depth (3).
- e. With the programmed feed through the centre of the groove back to point B (4).
- f. So the tool moves to and from, each time at another depth until the final depth is reached.



Depth movements in the groove.

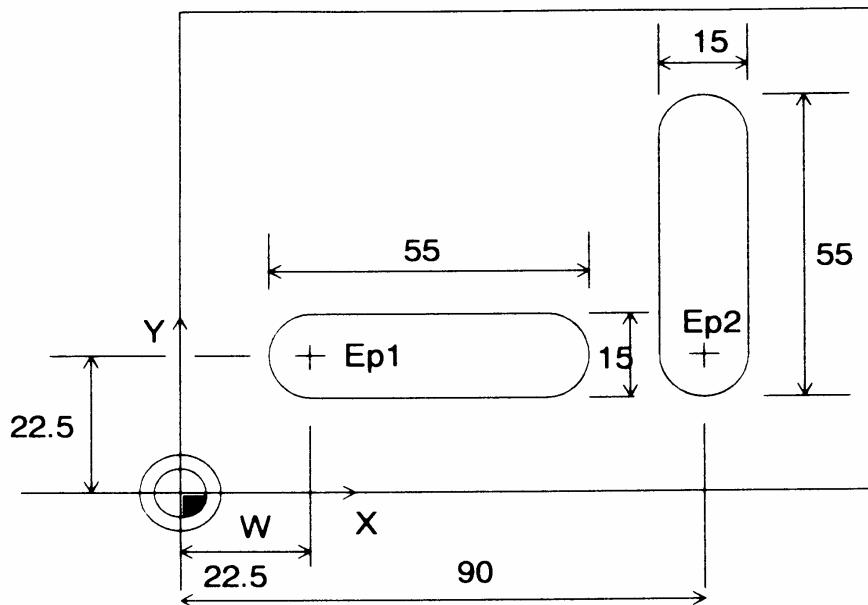
- g. Once the final depth is reached, the sides of the groove are milled from B to 1, 2, 3, 4, 1, and back to B in a counter clockwise direction or a clockwise direction if J-1 is programmed. Here the tool radius compensation is automatically activated by the control and cancelled when the cycle is completed. The radius of the tool is used with the radius compensation.



Cutter path for the sides.

- h. At the end of the cycle the tool is retracted out of the groove and stopped the clearance above the workpiece.

Example



N10 T1 M6 (Mill R5)	Load tool 1.	
N20 G88 X55 Y15 Z-5 B1 K1 Y3=10 F100 F2=200 S500 M3	Define groove milling cycle; parallel to X-axis.	
N30 G79 X22.5 Y22.5 Z0	Execute cycle at programmed position (Ep1).	
N40 G88 X15 Y55 Z-5 B1 K1 Y3=10 F2=200	Define groove milling cycle; parallel to Y-axis.	
N50 G79 X90 Y22.5 Z0	Execute cycle at programmed position (Ep2).	

Note: The F, S and M functions are still active and therefore need not to be programmed again.

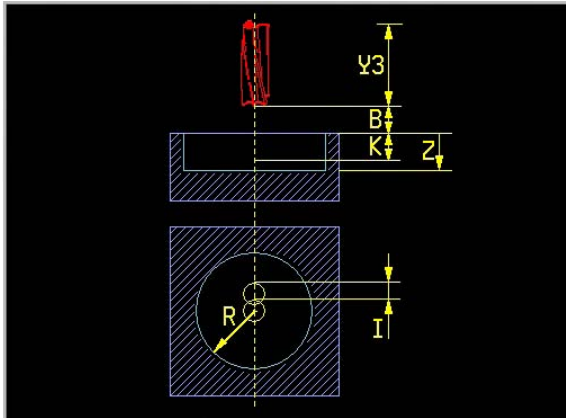
5.48 G89 Circular pocket milling cycle

To define in one program block the geometry of a circular pocket and some parameters for cutting the pocket.

Refer to the innovated cycle G789 and G799.

Format

G89 Z... R... {B...} {I...} {J...} {K...} {Y3=..} {F2=..}



G Circular pocket milling cycle
 Z Total pocket depth
 B Clearance
 I Cutting width mill in %
 J 1=climb, -1=conventional
 K Cutting depth
 R Radius circular pocket
 Y3= Special retract distance
 F2= In depth feed (only this block)

Pocket geometry

Z Total pocket depth
 R Radius circular pocket
 F2= In depth feed (only this block)

Cutting parameters

B Clearance
 I Cutting width mill in %
 J J1: climbing / J-1: conventional
 K Cutting depth
 Y3= Special retract distance

Notes and usage

Associated functions

G77, G79, G81, G83-G88

Total pocket depth (Z)

Total pocket depth measured from the surface. The sign of the Z-word indicates the direction of depth movement in the tool axis:

"-" in the negative direction, in most cases into the hole
 "+" in the positive direction.

PLANE OF OPERATION

The table below lists which axis in the active plane is used as the tool- AXIS.

	XY-PLANE	XZ-PLANE	YZ-PLANE
Z-word (tool axis)	Z-axis	Y-axis	X-axis

Feedrate to cutting depth (F2=)

Feedrate to cutting depth (F2=) is the feed of the depth movement in the toolaxis for each cleaning pass. If the Feed F2= is not defined in a cycle the Feed to the depth (K-word), is half the programmed feed (F).

Cutting width in % (I)

The value of the I-word states the percentage of the tool diameter to be used as cutting width for each cutting pass.

For example: I75 states that the cutting width is equal to 75% of the tool diameter.

If the I-word is not programmed, the value of a machine constant is (MC720).

Cutting depth (K)

If the pocket cannot be cleaned out at depth in one pass, the K-word is used to program the depth for each cleaning pass.

If the K-word is not programmed or has the same value as the Z-word, the pocket is milled in one pass.

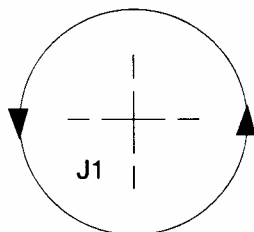
Special retract distance (Y3=)

The special retract distance Y3= is added to the clearance distance (B-word) this gives the opportunity to avoid objects.

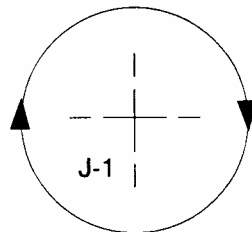
The special retract distance can have a sign, which will normally be the positive one.

If the special retract distance (Y3=) is not defined a retract movement is executed to the clearance distance.

Direction of milling (J)



J+1 Climb milling



J-1 Conventional milling

If no direction of milling is programmed, climb milling is assumed (J1= default direction).

Execution

A defined fixed cycle is executed on the position(s) programmed in either the G77 or G79 block. In these blocks the coordinates of the centre of the pocket are stated.

The pocket is milled in the active plane (G17, G18 or G19). The radius of the actual tool stored in the tool memory, is used for milling the pocket.

Fixed cycles should always be programmed with the G40-mode.

Cancellation

The cycle's values are cancelled when a new cycle is defined or by softkey CLEAR CONTROL, M30 or by softkey CANCEL PROGRAM.

Finishing the pocket

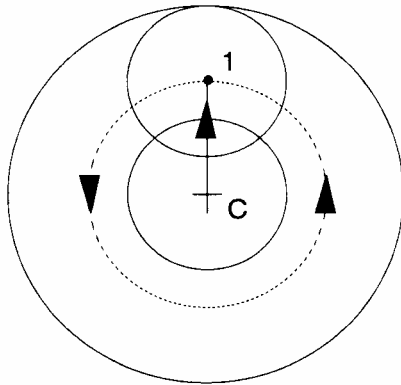
If a pocket must be milled and must have a finishing pass the following method may be used:

1. Add the allowance of the finishing cut to the tool radius and store the larger radius in the tool memory.
2. Execute the G89 milling cycle. A contour is milled which is smaller than the programmed contour due to the difference between the tool memory radius and the actual radius.
3. Program the contour of the pocket with the regular G1 and G2/G3 functions and execute the finishing pass with radius compensation and use the actual tool diameter.

Tool sequence

The tool sequence for milling the circular pocket is:

- With rapid to the centre (C) of the pocket and stay the clearance distance (B-word) above the workpiece
- With half the programmed feed to the first depth (K-word)

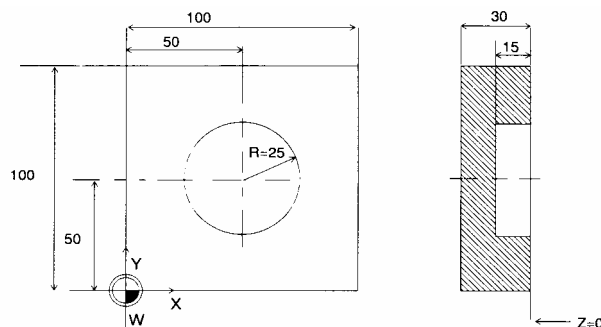


Tool sequence for a circular pocket.

- With the programmed feed from C to 1. The distance to move is: tool diameter x I-word.
- Make with the programmed feed a full circle in clockwise direction (J-1) or counter clockwise direction (J+1) as seen from the tool.
- The steps c and d are repeated until all material is cleaned out from the first layer.
- Go with three times the programmed feed back to point C.
- If the programmed depth is not reached, another movement over the depth (K-word) takes place and then another layer is cleaned out.
- If the total depth is reached, retract the tool out of the pocket and stop the clearance above the workpiece.

After cleaning out, a finishing for the side of the pocket might be necessary.

The best way to proceed is to store in the tool memory for the actual tool a radius, being the stock removal greater than the actual radius of the tool. Once the cycle is totally executed this stock removal remains for finishing. Activating the tool radius compensation and using the regular G1 and G2/G3-blocks program the finishing of the pocket.

Example

N10 T1 M6 (Mill R5)

N20 G89 Z-15 B1 R25 I75 K6 F200 S500 M3

N30 G79 X50 Y50 Z0

N40 G0 Z200

Load tool.

Define the cycle for milling a circular pocket.

Execute cycle at programmed position.

Retract the tool

5.49 G90/G91 Absolute/incremental programming

Absolute and incremental programming can be done on two ways:

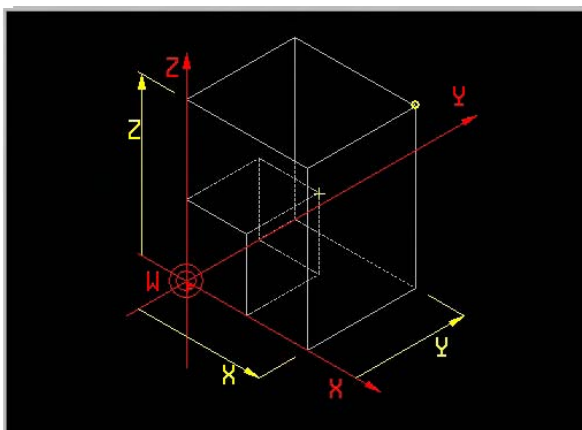
- 1) Absolute and incremental programming with G90 and G91.
- 2) Wordwise absolute and incremental programming with X90= and X91=.

5.49.1 G90/G91 Absolute/incremental programming

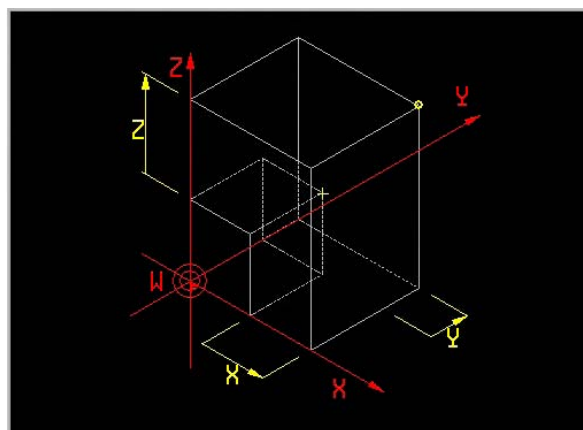
To select one of two modes of coordinate programming.

Format

G90/G91 [Axis coordinates]



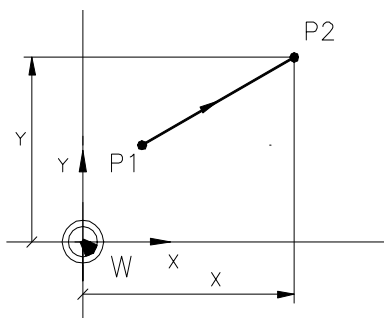
G90



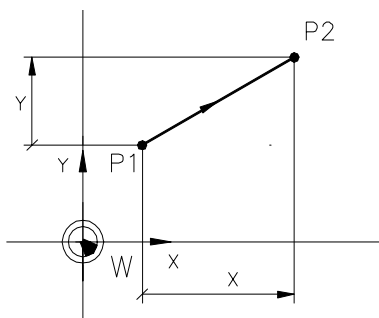
G91

G90: Absolute coordinates measured from program zero point W.

G91: Incremental coordinates measured from last programmed tool position.



G90: Absolute coordinates



G91: Incremental coordinates

Notes and usage

Modality

G90 and G91 are modal functions.

Possible axes coordinates by G90/G91 are:

X, Y, Z	Endpoint coordinates
A, B, C	End angels

Default mode

The G90 absolute coordinate mode automatically becomes active when the CNC system is switched on or at CLEAR CONTROL.

Cancellation

The G91 can be cancelled when G90 is defined or by softkey CLEAR CONTROL, M30 or by softkey CANCEL PROGRAM.

Switching between the two modes

The G90 default mode is cancelled by programming G91. The coordinates, which follow the switch, are interpreted by the CNC as incremental coordinates. To reactivate the absolute mode, G90 has to be programmed.

Internally the control operates with absolute coordinates. Therefore, within a particular program it is possible to change arbitrarily from absolute to incremental and vice versa.

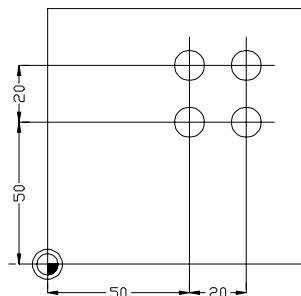
Polar coordinates

The polar coordinates (B1=, L1=), (B2=, L2=), (B3=, L3=) are not influenced by the functions G90 and G91. They can be used arbitrarily in each coordinate mode.

Position display

The axis positions on the display of the control are always absolute coordinates and related to the program zero point W.

Note: A part program should always contain an absolute position, before G91 is used. If a program starts immediately with G91, the actual tool position is used as the first absolute position of the program. Each time the program runs, this position should be the same, otherwise the program is executed each time at another place.

Example

```

N88550
N1 G17
N2 G54
N3 G195 X0 Y0 Z60 I100 J100 K-80
N4 S1300 T1 M6 (Drill R5)
N5 G0 X0 Y0 Z50
:
N8 G81 Y2 Z-10 F200 M3
N9 G79 X50 Y50 Z0
N10 G91
N11 G79 Y20
N12 G79 X20
N13 G79 Y-20
N14 G90
:
N17 G0 X0 Y0 Z50 M30

```

```

Set the plane to be the XY-plane
Set the zero point
Graphic window definition
activate Tool 1
Move tool with rapid speed to start position

Drilling cycle definition
Drilling the first hole
Switching to incremental mode
A incremental movement to the second hole
A incremental movement to the third hole
A incremental movement to the third hole
Switching to absolute mode

Retract tool End of program

```

5.49.2 Wordwise absolute and incremental programming

Wordwise absolute and incremental programming, independently of G90/G91.

Format

Programming absolute:

G.. [Axisname]90=...

Programming incremental:

G.. [Axisname]91=...

Notes and usage**Parameters**

Axisname: X, Y, Z, I, J, K, A, B, C

X90=, Y90=, Z90= Absolute endpoint

A90=, B90=, C90= Absolute endpoint angle

X91=, Y91=, Z91= Incremental endpoint

A91=, B91=, C91= Incremental endpoint angle

Associated functions

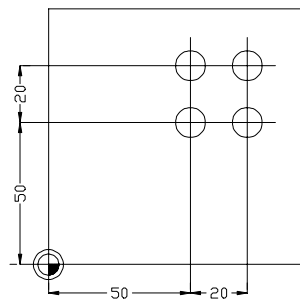
G0, G1, G2, G3, G9, G45, G46, G61, G62, G77, G79, G145, G182

Cartesian coordinates:

The wordwise absolute and incremental programming does not depend on the modally valid system of measurement 90/G91.

Polar coordinates:

Programming with polar coordinates will not be influence.

Example

N88550

N1 G17

N2 G54

N3 G195 X0 Y0 Z60 I100 J100 K-80

N4 S1300 T1 M6 (Drill R5)

N5 G0 X0 Y0 Z50

Set the plane to be the XY-plane

Set the zero point

Graphic window definition

activate Tool 1

Move tool with rapid speed to start position

N8 G81 Y2 Z-10 F200 M3

N9 G79 X50 Y50 Z0

N11 G79 Y91=20

N12 G79 X91=20

N13 G79 Y91=-20

:

N17 G0 X0 Y0 Z50 M30

Drilling cycle definition

Drilling the first hole

Incremental Y movement to the second hole

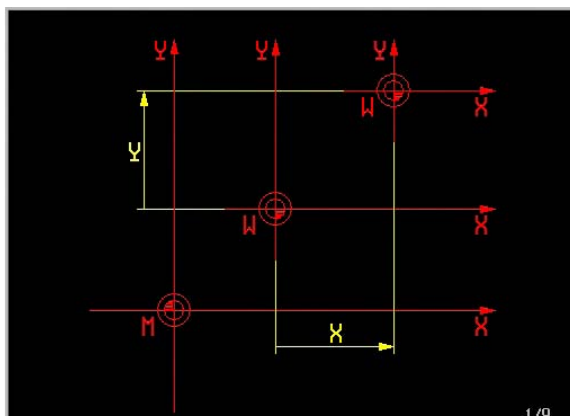
Incremental X movement to the third hole

Incremental Y movement to the third hole

Retract tool and end of program

5.50 G92/G93 Incremental/Absolute zero point shift

G92



G Zero point shift incr./ rotation
 X Zero point coordinate
 Y Zero point coordinate
 Z Zero point coordinate
 B Zero point angle
 C Zero point angle
 B1= Angle
 B4= Angle of rotation incremental
 L1= Path length

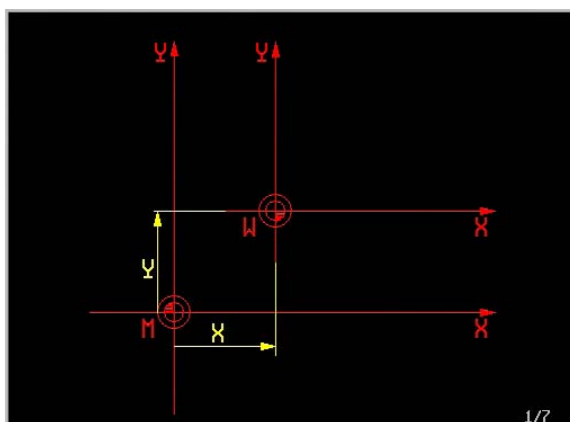
Words used with G92

1. Zero point shift

X, Y, Z	Zero point coordinate
A, B, C	Zero point angle
B1=	Angle
L1=	Path length
2. Axis rotation

B4=	Angle of rotation incremental
-----	-------------------------------

G93



G Zero point shift abs./ rotation
 X Zero point coordinate
 Y Zero point coordinate
 Z Zero point coordinate
 B Zero point angle
 C Zero point angle
 B2= Polar angle
 B3= 1=Reset position 0-360 degrees
 B4= Angle of rotation absolute
 L2= Polar length
 P1= Point definition number
 C3= 1=Reset position 0-360 degrees

Words used with G93

1. Zero point shift

X, Y, Z	Zero point coordinate
A, B, C	Zero point angle
B2=	Polar angle
L2=	Polar length
P, P1=	Point definition number
2. Axis rotation

B4=	Angle of rotation incremental
-----	-------------------------------
3. Reset function

A3=, B3=, C3=	reset parameters
---------------	------------------

With G93 A3=1 the corresponding rotary axis position is reset to a value between 0 and 360 degrees. An A-axis with the position 370 degrees is changed after the programming of G93 A3=1 to 10 degrees.

Format

Zero point shift

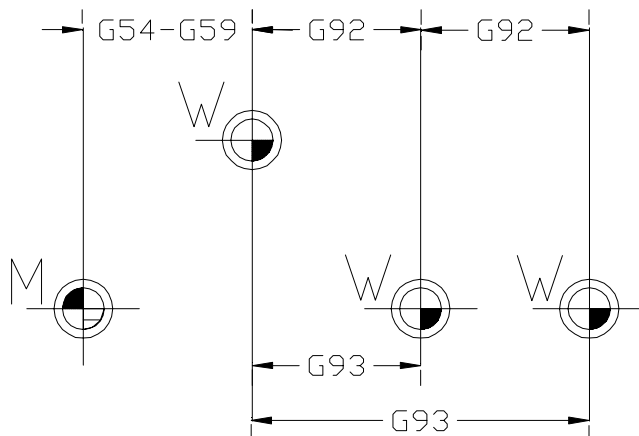
G92 [Coordinate(s) related to last zero point]

G93 [Coordinate(s) related to a fixed zero point]

Axis rotation

G92/G93 B4=...

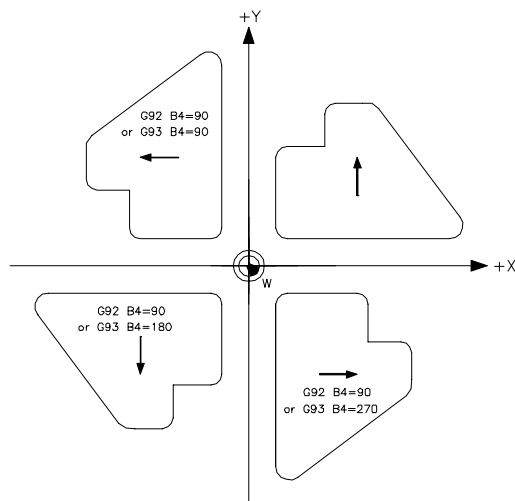
- To establish a program zero point (W) on the workpiece so that workpiece dimensions can be used directly for programming tool or workpiece movements. (G54-G59 or G54I[nr.])



G92: Incremental zero point shift; shift values are related to last program zero point.

G93: Absolute zero point shift; shift values are related to a fixed zero point

- To produce a rotation of coordinate axes for a group of coordinates



G92: Rotation angle related to the last programmed main axis

G93: Rotation angle related to a fixed machine tool.

Notes and usage**Modality**

G92 and G93 are modal functions.

Associated functions

With zero point shift: G51/G52, G53-G59 or G54I

With axis rotation: G72/G73

Zero point shifts**Incremental zero point shift (G92)**

The function G92 is used to shift the zero point from the actual program zero point (W) to a new program zero point (W).

Application of G92

The function G92 is useful when programming identical tool movements, which are repeated at different locations on a workpiece.

Absolute zero point shift (G93)

The function G93 is used to establish the program zero point by shifting the zero point from the mounting zero point (C) to the required program zero point (W).

Unaltered axis

When a zero point shift is made and an axis is not involved, that axis does not need to be included in the block where the shift is programmed.

Programmed dimensions

All programmed dimensions which follow a zero point shift are measured from the new zero point (0,0).

Displayed coordinates

Displayed axis coordinates are always related to the active program zero point W.

Cancellation of G92 and G93

With the G92 zero point shift is only the programmed addresses added to G93. With the G93 zero point shift is only the programmed addresses overwritten to a G92.

G93 X....

G92 Y.... Y is added to the G93 zero point shift.

A programmed zero point shift (G92 or G93) is cancelled if another zero point shift function (G51/G52, G53-G59 or G54I[nr.]) is programmed.

A programmed zero point shift (G92 or G93) is cancelled at end of program or by softkey CLEAR CONTROL, M30 or by softkey CANCEL PROGRAM.

Rotation of axes

The main plane axes can be rotated around the program zero point W. In this way a part program or a section of a part program can be rotated.

The programmed Coordinates refer to the rotated axes.

Plane selection

Axis rotation is performed in the active main plane, thus:

- G17: X- and Y-axis are rotated
- G18: X- and Z-axis are rotated
- G19: Y- and Z-axis are rotated

Angle of rotation

The angle of rotation is programmed with the word B4=.. The angle ranges from -360° to 360° and is measured as with polar coordinates.

Incremental angle of rotation (G92)

With G92 the angle is measured with the last active coordinate axis:

- with G17 or G18: the (rotated) X-axis
- with G19: the (rotated) Z-axis

Absolute angle of rotation (G93)

With G93 the angle is measured with the machine tool axis:

- with G17 or G18: the fixed X-axis
- with G19: the fixed Z-axis

Zero point shift and axis rotation

In a G92 or G93 block a zero point shift and axis rotation is allowed. The order of execution is:

- first the zero point shift,
- then axis rotation.

The new zero point is the centre of rotation.

G51-G59 zero point shifts

If one of the G-functions G51 to G59 is programmed after axis rotation, the function is executed in the non rotated axes.

Mirror image and scaling

A combination of mirror image and/or scaling and axis rotation is allowed. The order of execution is:

- first scaling and mirror image,
- then axis rotation.

Displayed coordinates

After axis rotation the displayed axis coordinates are related to the non rotated axes of the main plane.

Cancellation

The G92 axis rotation is cancelled, if a G93 axis rotation is programmed.

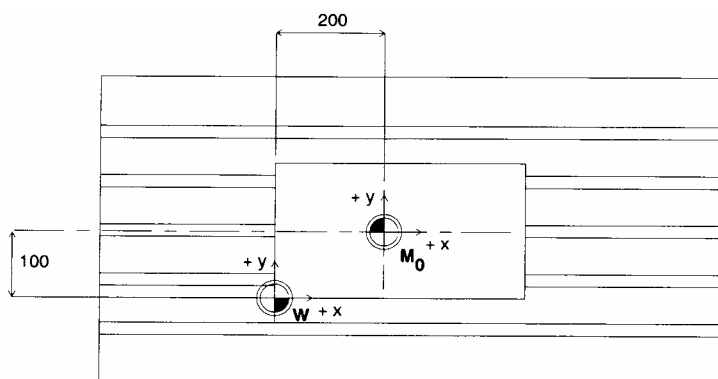
The G92 axis rotation is added to a G93 axis rotation.

The G93 axis rotation is cancelled with G93 B4=0.

Both axis rotations (G92 and G93) are cancelled at end of program or by softkey CLEAR CONTROL, M30 or by CANCEL PROGRAM.

Examples

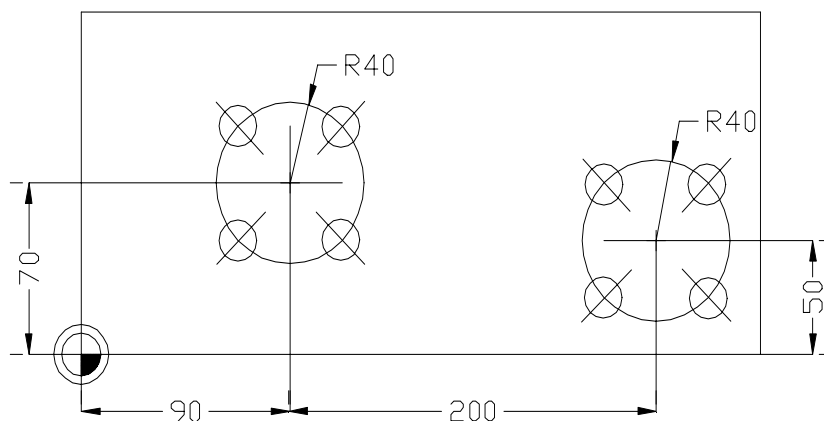
Example 1.



In this example the centre of the workpiece coincides with the machine zero point (M_0), and the program zero point (W) is to be located in the left hand corner of the workpiece. A G93 can be used to set the program zero point:

```
N30 G93 X-200 Y-100
```

2. Example 2



The four holes around point A and the four holes around B are to be drilled. In the program the program zero point (W) is located at A or at B. In this way calculations during programming are reduced to a minimum.

Program with G92

```
N79560
```

```
N1 G17
```

```
N2 G54
```

```
N3 G195 X-10 Y-10 Z10 I420 J180 K-30
```

```
N4 G99 X0 Y0 Z0 I420 J160 K-10
```

```
N5 F200 S3000 T1 M6
```

```
N6 G92 X90 Y70
```

```
N7 G81 Y1 Z-12 M3
```

```
N8 G77 X0 Y0 Z0 I45 J4 R40
```

```
N9 G92 X200 Y-20
```

```
N10 G14 N1=8
```

```
N11 G92 X-290 Y-50
```

```
N12 G0 Z100 M30
```

Set the plane to be the XY-plane

Set the zero point

Graphic window definition

Define the blank of the workpiece as a box

activate Tool 1

Incremental zero point shift

Drilling cycle definition

Drilling the four holes on the circle

Incremental zero point shift

Repeat program block 8. Drilling the four holes on the circle

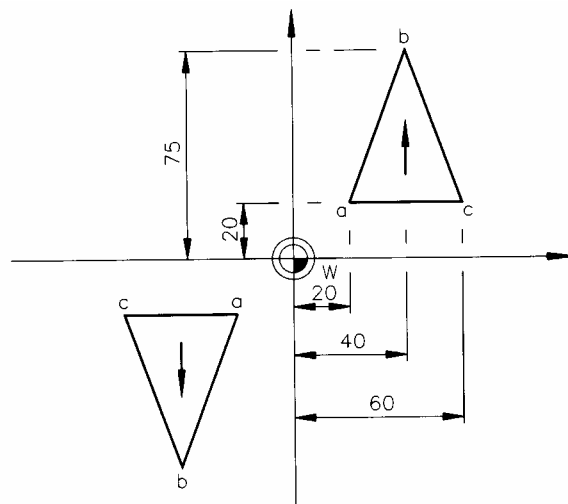
Incremental zero point shift back to the first zero point shift

End of program

Program with G93

Related to the mounting point C, the program looks as follows:

N79561	
N1 G17	Set the plane to be the XY-plane
N2 G54	Set the zero point
N3 G195 X-10 Y-10 Z10 I420 J180 K-30	Graphic window definition
N4 G99 X0 Y0 Z0 I420 J160 K-10	Define the blank of the workpiece as a box
N5 F200 S3000 T1 M6	activate Tool 1
N6 G93 X90 Y70	Absolute zero point shift
N7 G81 Y1 Z-12 M3	Drilling cycle definition
N8 G77 X0 Y0 Z0 I45 J4 R40	Drilling the four holes on the circle. The surface of the workpiece is define as Z=0.
N9 G93 X290 Y50	Absolute zero point shift
N10 G14 N1=8	Repeat programblock 8. Drilling the four holes on the circle
N11 G93 X0 Y0	Absolute zero point shift back to the first zero point shift
N12 G0 Z100 M30	End of program

Example 3 Rotation of axes

N9300	
N1 G17	Set the XY-plane
N2 G54	Set the stored zero offset
N3 S400 T1 M6	Load tool 1 and set the spindle speed
N4 G0 X60 Y-10 Z1 M3	Move tool to start position. Make spindle rotate clockwise at 400 rev/min
N5 G1 Z-16 F1000	Move tool to depth at set feedrate
N6 G43 Y20	Move tool to point C
N7 G41	Set radius compensation LEFT and cut workpiece
N8 G1 X20	
N9 X40 Y75	
N10 X60 Y20	
N11 Y-10	
N12 G40	Cancel radius compensation
N13 G0 Z10	Move tool rapidly out of workpiece
N14 G93 B4=180	Rotate axes through 180 degrees
N15 G14 N1=4 N2=12	Repeat instructions given in block N4 to N12
N16 G0 Z100	Retract the tool from the part
N17 M30	Cancel rotation of axes and end of program

5.51 G94/G95 Select feedrate unit

To control how the CNC interprets programmed (F-word) feedrate values.

By turning mode refer to paragraph 12.5 "Extended select feedrate unit" on page **Error! Bookmark not defined..**

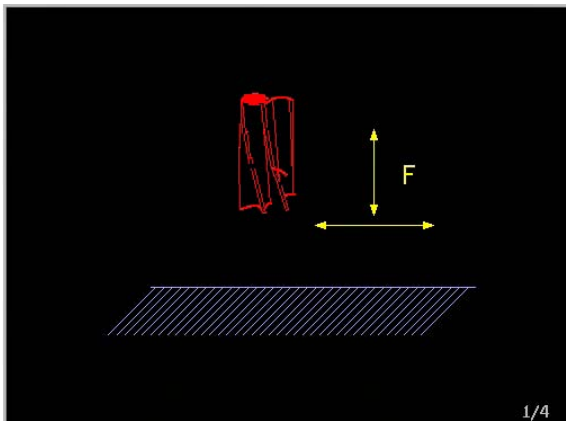
Format

G94/G95 F...

G94 F5=.

F5=0 degrees/min (default)

F5=1 mm/min or inches/min



```
G Feed in mm/min (inch/min)
F Feed
F1= F-adaptation:1=red.,2=r/h,3=high
F3= In depth feed
F4= In plane feed
F5= Feed rotary Axes
```

G94

```
G Feed in mm/rev (inch/rev)
F Feed
F1= F-adaptation:1=red.,2=r/h,3=high
F3= In depth feed
F4= In plane feed
```

G95

Notes and usage

Modality

G94 and G95 are modal functions.

Default mode

G94 is automatically made active when the CNC is switched on or by softkey CLEAR CONTROL, M30 or by CANCEL PROGRAM.

Dimensional unit

The dimensional unit for both functions is determined by the functions G71 (metric) or G70 (inches).

Conversion to a feedrate in units/min

When G95 is active the CNC automatically converts the F-value to a feedrate in mm/min (inches/min).
If a spindle transducer is fitted to the machine, the measured spindle speed is used for this calculation.

Machines with kinematic model

The function G94 F5= is only possible if a kinematical model is defined for the machine. (MC312 must be active).

Rotary axis radius calculation G94 F5=1

In machines with the kinematical model the rotary axis radius between the centre point of the rotary axis and of the workpiece can be calculated. This means that A40=, B40= and C40= no longer need to be programmed.

Turn off G94 F5=1

G94 F5=1 is cancelled by G94 F5=0, G95, the programming with A40=, B40= or C40= in G0 or G1, M30, <program abort> or <reset CNC>.

Example

N.. G94

N.. G1 X.. Y.. F200

The tool is moved to a point defined by the coordinates X.. and Y.. at a feedrate of 200 mm/min

N.. G95

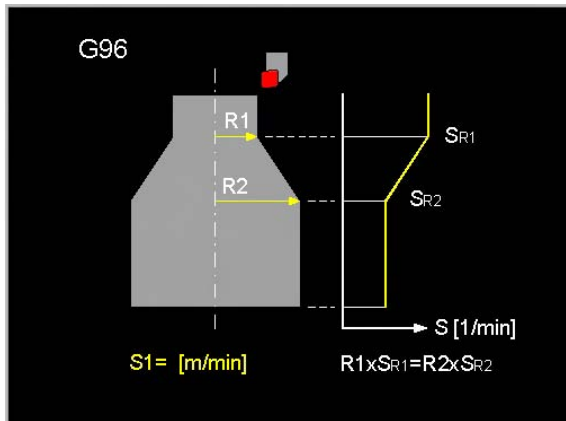
N.. G1 X.. Y.. F.5

The tool is moved to a point defined by the coordinates X.. and Y.. at a feedrate of .5 mm/rev.

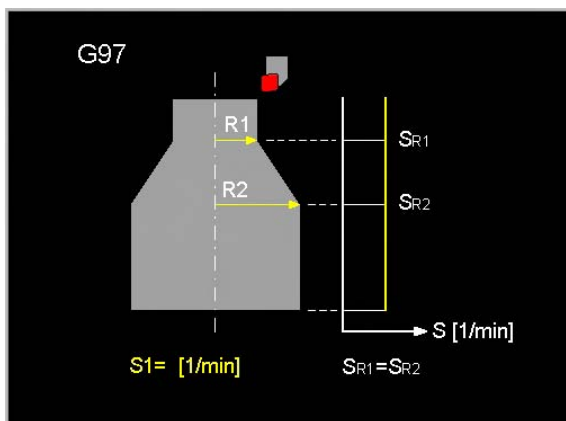
5.52 G96/G97 Constant cutting speed

G96 Programming constant cutting speed.

G97 switches off constant cutting speed.



G Constant cutting speed
 D Upper speed limit (rev/min)
 F Feed
 S Cutting speed (m(feet)/min)
 M Machine function
 S1= Cutting speed (m(feet)/min)
 M1= Machine function



G Spindle speed
 S Speed (rev/min)
 M Machine function
 S1= Speed (rev/min)
 M1= Machine function

Refer to chapter "Turning mode".

5.53 G98 Graphic window definition

Remark

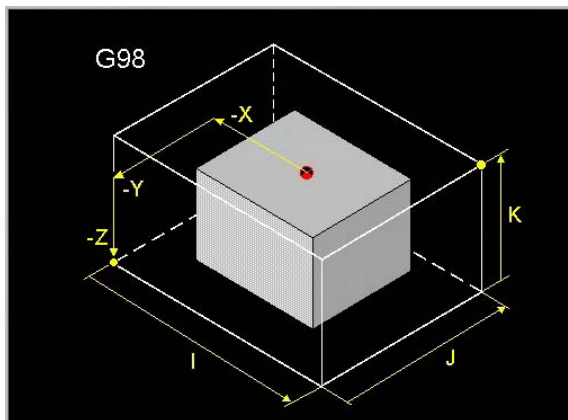
It is recommended to start a new program always with G195.

Purpose

To define the position relative to the zero point W and the dimensions of a 3D graphics window in which the machining of a workpiece is to be represented with the graphical simulation of a part program run on the display of the control.

Format

G98 X... Y... Z... I... J... K... {B...} {B1=...} {B2=...}



```
G  Graphic window definition
X  Start point coordinate
Y  Start point coordinate
Z  Start point coordinate
B  Rotation around hor. axis (3D)
I  Dimension parallel to X
J  Dimension parallel to Y
K  Dimension parallel to Z
B1= Rotation around vert. axis (3D)
B2= Rotation around third axis (3D)
```

Notes and usage

Associated functions

G99, G195 to G199

Graphical support

Refer to the appendix GRAPHICAL SUPPORT at the end of this manual for a short overview about the graphical support provided in the CNC PILOT control system and to the user manual for using the graphical support.

Graphic window

The window, thus a bounded area on the display, is a rectangular 3D box which dimensions are defined by the G98-function. The dimensions of the workpiece are defined in a G99 block.

The window is used with the graphical simulation, but also with the synchron graphics with which the actual tool movements on the machine can simultaneously be seen on the display of the control.

Default window dimensions

If the 3D window dimensions are not defined the CNC uses the limit switches distances as default values.

Tool image

A tool image can be assigned to the tool with the aid of the G-word in the tool memory. The required image can be selected from a set of available tool images and is used by the CNC system to accurately simulate the machining.

(For tool image refer to operation manual)

Angle of viewing (B, B1=, B2=)

With the simulation graphics or the 3D-wire plot the workpiece can be seen rotated. The angles for viewing the rotated workpiece on the display are defined by the words B, B1= or B2=.

	B	B1=	B2=
	rotation about	rotation about	rotation about
XY-plane (G17)	X-axis	Y-axis	Z-axis
XZ-plane (G18)	Z-axis	X-axis	Y-axis
YZ-plane (G19)	Y-axis	Z-axis	X-axis

Other methods are available for selecting an angle of viewing and are described in the user manual.

Default settings for angles of viewing

If the angles of viewing are not programmed the following default settings are automatically used by the control: B60, B1=30, B2=0

Note The function G98 must be programmed before G99.
It is recommended to start a new program always with G195.

Example

```

N9000
N1 G98 X-20 Y20 Z-25 I140 J-90 K85   Define the start point and dimensions of the 3D graphic
                                     window.
N2 G99 X-15 Y15 Z-20 I130 J-80 K75   Define the blank of the workpiece as a box.

```

5.54 G99 Definition of workpiece blank as a box

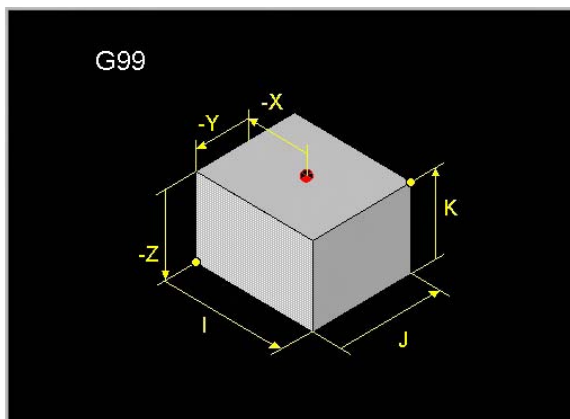
Remark

It is recommended to start a new program always with G199.

To define the dimensions of a 3D box used as a blank (uncut) workpiece and the position of this 'blank' relative to the program zero point W. These dimensions are used in a graphical simulation of a partprogram run. (This function is used together with the G98-function).

Format

G99 X... Y... Z... I... J... K...



G	Graphic: material definition
X	Start point coordinate
Y	Start point coordinate
Z	Start point coordinate
I	Dimension parallel to X
J	Dimension parallel to Y
K	Dimension parallel to Z

Notes and usage

Associated functions

G98, G196 to G199

Irregular workpiece shape

If the blank of the work piece cannot be defined with one box, the functions G196 to G199 must be used to define a workpiece, which has an irregular shape.

Restriction

Only one G99 function is allowed in a part program.

Availability of the G99 function enables the part programs developed for the CNC 3000 to be used on the CNC Pilot as well. Unlike the CNC 3000, the program of the CNC Pilot can only contain a single G99 function. If a particular CNC 3000 program includes several G99 functions, the contour should be reprogrammed, using the functions G196 up to G199.

Note The function G98 must be programmed before G99.
It is recommended to start a new program always with the functions G196 till G199.

Example

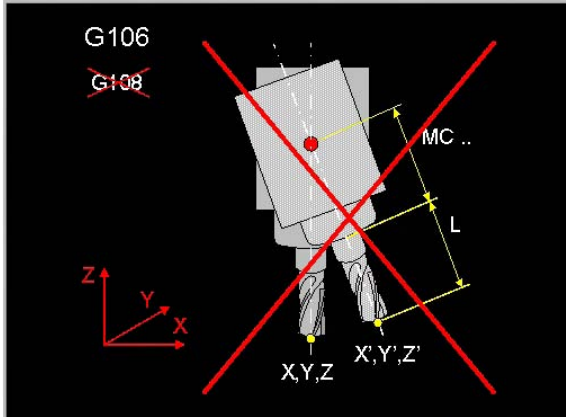
N9000	
N1 G98 X-20 Y-20 Z20 I140 J90 K-85	Define the dimensions of 3D graphical window.
N2 G99 X0 Y0 Z0 I100 J50 K-55	Define the blank as a box

5.55 G106 Kinematic Calculation: OFF

Switches off G108 (Calculate kinematics: ON).

Format

G106



G Kinematic calculation: off

Notes and application

Modality

This function is modal with G108.

Execution

G106 waits with all actions until the movement in the preceding block is finished with <INPOD>. G106 deactivates calculation of the kinematics. The active offset in the linear axes is cancelled.

Note: G106 has the same effect as G108 I1=0 or MC756=0 (no calculation of kinematics).

Display

The G106/G108 functions remain in processing status in the modal G series. There is no separate symbol (as with G7/G8/G141) for the status with G108 active.

Example

N10 G106

Switch off G108.

5.56 G108 Kinematic calculation: ON

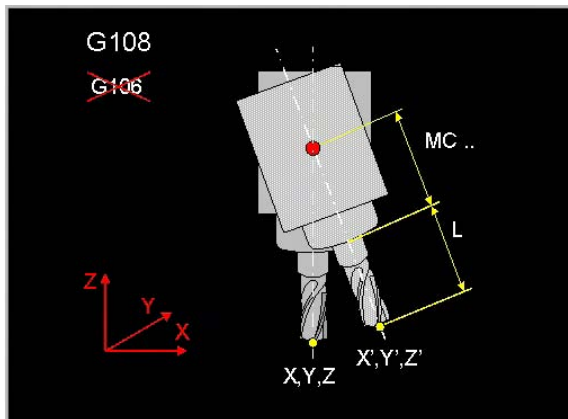
Function whereby, with rotated circular axes, the position of the tool tip is calculated using the kinematic model. G108 activates calculation of the kinematics.

The status of the tool head is calculated at the end of a positioning movement into the position of the linear axes. The linear axes are not included.

The position display of MillPlus **IT** takes account of a change in the machine kinematics, such as would occur when a head is tilted. The offset caused is compensated for by an absolute programmed movement of the axes concerned.

Format

G108 {I1=...}



```
G Kinematic calculation: on
I1= Kinematic {0,1=head+tool,2=head}
```

I1= 0 = same as G106
 1 = tool head and tool length is compensated
 2 = only tool head is compensated

Basic settings

Depending on MC756. This setting is active again after <Clear Control> and M30

If G108 is programmed without parameter, I1=1 is default

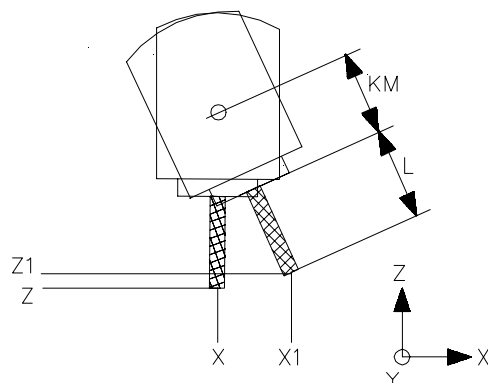
Notes and application

Modality

This function is modal with G106.

Execution

G108 waits with all actions until the movement in the preceding block is finished with <INPOD>.



KM = calculation with the kinematic model.

X, Z is the starting position. Tool length compensation is calculated in the Z direction.

X1, Z1 is the display position when G108. The head position is calculated in the rotated direction and if I1=1, tool length compensation is calculated in the Z direction (depending on G17).

Warning: If G108 is active, the position of the tool tip in intermediate positions of this rotary axis is different from what it was previously (The PLC program has been adapted for this and the calculation is no longer compatible).

Warning: This could make existing NC programs cause collisions.
If G108 is calculating the tool length (I2=1) the tool direction is no longer defined by G17/G18/G19 or G66/G67.
This could make existing NC programs cause collisions.

Switch off G108

G106 switches the G108 function off. G108 is reactivated in the MC basic setting (MC756) after <Program Cancel>, M30 <Clear Control> or switching on the CNC.

Machine zero point

It is assumed in the function G108 that the zero point is defined in the vertical position of the tool head. In the horizontal position (or in-between positions the position is corrected).

Rotary axis movement

When G108 is active the linear axis display is updated at the end of every positioning movement of the rotary axes defined in G108. <INPOD> then rapidly stops movement.

Interruption

When a rotary axis movement is interrupted the linear axis display is not updated. During an interruption the linear axis display is only updated to show the rotary axis status after <Emergency stop>, <Cancel program> or <Manual> has been pressed.

Manual

The G108 function remains active after M30 and is active during manual operation. The linear axis display is updated when rotary axis movement stops.

Kinematic model

The function is active for all machine tool types with rotary axes in the tool head.

Machine constants

MC 756 Calculate Kinematics (0 = no, 1 = with tool length, 2=without tool length)

Defines whether the function G108 is activated automatically after switching on the CNC and <Clear Control> and M30. With G108 is defined whether the rotary axes positions are processed in the display of the linear axes.

0 = G106 is active after switching on

G108 can be programmed, but after <Program Cancel> or M30 G106 is active again.

1 = G108 is active after switching on. The rotary axes in the tool head and the tool length are processed in the kinematic model.

2 = G108 is active after switching on. The rotary axes in the tool head are processed in the kinematic model.

Warning: When MC756 is activated existing NC programs could cause collisions.

Example Kinematic model permanently active.

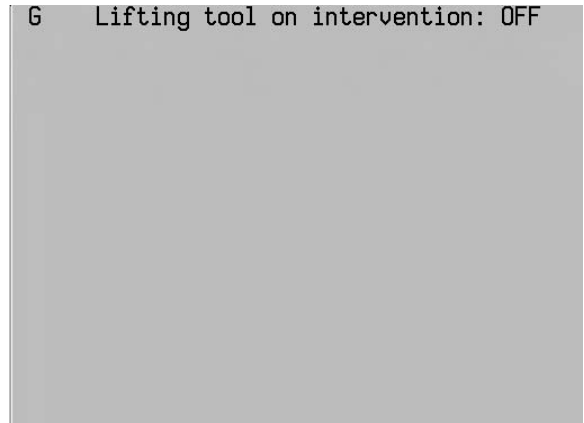
Program example	Description
N10 G108	Calculation of rotary axes in the tool head.

5.57 G125 Lifting tool on intervention: OFF

Deactivating the tool lifting movement.

Format

G125



Notes and application

Modality

This function is modal with G126

Execution

G125 resets the modal <Tool lifting enabled status> of the G126 function. After this no tool lifting movement can occur.

G125 is identical to G126 I1=0 I2=0 I3=0

G125 causes <INPOD>.

Display

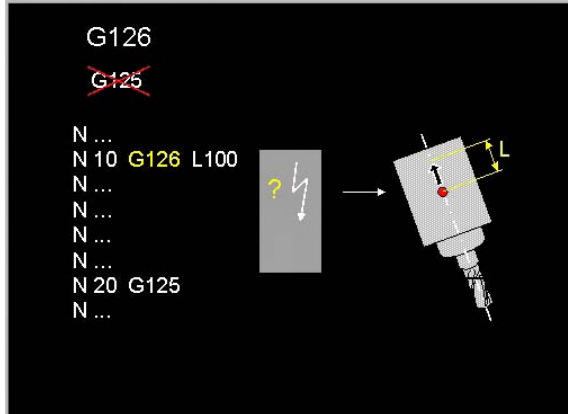
The function G125/G126 are listed in the modal G-group in the operating status.

5.58 G126 Lifting tool on intervention: ON

G126 is a function to lift the tool from the work piece under certain conditions (coolant failure, intervention and errors).

Format

G126 {I1=..} {I2=..} {I3=..} {L..}



G Lifting tool on intervention: ON
 L Lifting distance in tool dir.
 I1= Lifting by PLC: 0=off,1=on
 I2= Lifting on <INT>: 0=off,1=on
 I3= Lifting on error: 0=off,1=on

- I1= Tool lifting by PLC (Coolant failure): 0= no lifting, 1= lifting
- I2= Tool lifting at intervention <INT>: 0= no lifting, 1= lifting
- I3= Tool lifting at errors: 0= no lifting, 1= lifting
- L= Lifting distance in the tool direction
- L Defines the distance in the tool direction or tool orientation direction (G36 turning) over which is lifted. Default value in 'MC758 tool lifting distance'.
 Value between 0.001 and 99999.999 [mm] or 0.0001 and 9999.9999 [inch]

Basic settings

I1=1, I2=0, I3=0, L=MC758

Notes and application

Modality

This function is modal with G125.

Execution

G126 causes <INPOD>. After this a modal <tool lifting enabled status> is set.

The tool lifting movement is activated when:

- An event as described in I1 - I3 (coolant failure, intervention or error) occurs.
- The G126 Modal <tool lifting enabled status> is activated.
- A feed is active. In case the feed override is set to zero, no tool lifting takes place.
- During fixed cycles also when rapid is active.
- Certain G functions are activated.

Remark: Also when the tool lifting movement was not activated, the movement stops. When e.g. WOX_RETRACT_TOOL is set during rapid, the movement stops without a tool lifting movement.

The tool lifting movement occurs:

- in the programmed direction
- in the tool direction (G37 'milling', G126 L parameter or basic setting), or until the programmed tool lifting height or the SW end switch is reached.

After the tool lifting movement, the program execution and the spindle is stopped with an (additional) error message 'I264 Machining stopped with lifted tool'.

Remark: When the tool lifting movement is activated by an error (G126 I3=1) which also causes emergency stop, the servo's are already switched off before the tool lifting movement has ended.

Movement sequence

Before the tool lifting movement starts, the MillPlus decelerates until the correct (jerk free) angle velocity is reached.

During the following G functions, even when the G126 function is active, the tool lifting movement is not possible:

Movements	0, 6, 31, 33	Depending on the G28 setting for the feed movements
Planes	7, 182	
Measuring cycles	45, 46, 49, 50, 145, 148, 149, 150	
Positioning	74, 174	
Fixed cycles	84, 86	
New cycles	784, 786, 790, 794	
Graphics	98, 99, 195, 196, 197, 198, 199	
Pocket cycle	200, 201, 203, 204, 205, 206, 207, 208	

Switching off G126

At <M30>, <Program cancel>, G125 active and <Clear control> G126 '(Tool lifting on intervention: ON)' is deactivated.

Status display

The G125 / G126 status is shown in the modal G-group display.

Manual block search

During manual block search the functions G125 and G126 are maintained. The last one is executed before repositioning and output.

Interrupt of the tool lifting movement

The tool lifting movement itself can be interrupted. However, after interruption it is not completed. A new <Start> causes repositioning.

Repositioning

After the tool lifting movement the normal possibilities during intervention are available. Repositioning occurs with positioning logic.

Machine constants

MC 756 Tool lifting movement distance
Value between 1 and 99999999 [um].

With G320 the status of G126/G125 and the programmed distance can be requested:

I1=72 Programmed status
 0 = G125
 1 = PLC (G126 I1=1)
 2 = INT (G126 I2=1)
 3 = PLC + INT (G126 I1=1 I2=1)
 4 = ERR (G126 I3=1)
 5 = PLC + ERR (G126 I1=1 I3=1)
 6 = INT + ERR (G126 I2=1 I3=1)
 7 = all (G126 I1=1 I2=1 I3=1)
 I1=73 Programmed distance

Example Activate tool lifting function.

Programming example	Description
N10 G126 I1=1 I2=1	Activating the tool lifting function by IPLC or intervention.

5.59 G136 Second axes configuration for fork head: ON

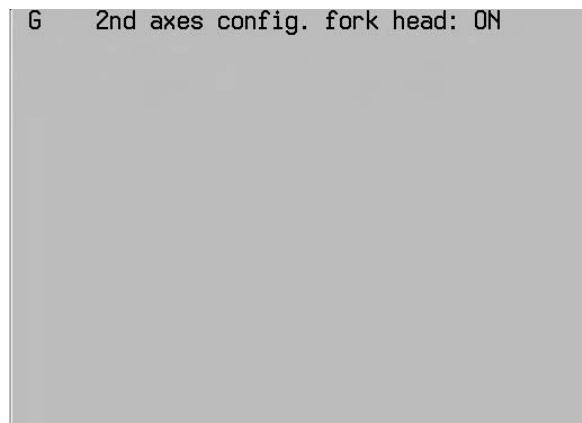
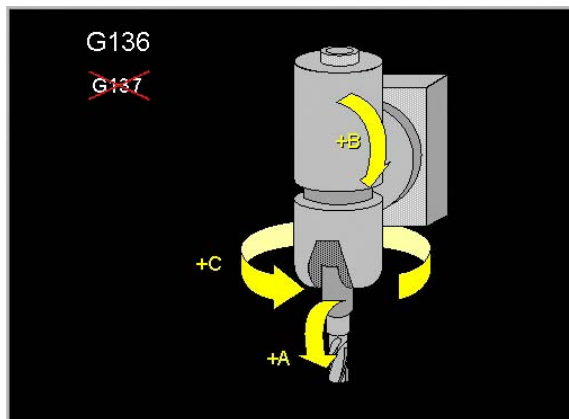
With G136 a -by the machine tool builder fixed implemented- function is activated (**e.g. a fork head moved into position**) Doing so a second axes configuration is activated. See the machine tool manual for the possibilities. In case your machine tool is not equipped with such kind of device the functions G136 and G137 have no meaning.

General description of the moveable fork head

The machine tool is delivered with a moveable fork head. In this case the machine tool has two configurations:

- 1 Normal head
- 2 fork head

With a continuous controlled fork head (B-axis, second C-axis and A-axis) it is possible to machine surfaces with five axes.



Moving the fork head into position must be started by an M-function (see machine tool manual). By activating the moveable fork head by G136, the main axis C (rotary table) is exchanged with the fourth auxiliary axis. The fourth auxiliary axis controls the C-axis in the fork head. The fork head, activated by G136 is de-activated by G137 and the C-axis is changed back from C-axis-head to C-axis-table.

Actions when using the fork head:

- 1 Output of an M-function (defined in MC_1063) to move the fork head into position. The kinematic model, defined by the machine tool builder is exchanged.
- 2 Output of the G-function (G136) to activate the fork head. The C-axis in the table is exchanged with the C-axis in the head.

Example: activating the fork head

In this example is assumed that M153 and M154 are used to move the fork head into position:

M153: Move the normal head into position (default)

M154: Move the fork head into position

Program example	Description
N9000 (smart fräsen)	
N10 G17	Select plane XY
N20 G7	Switch off G7
N30 M153	Move the normal head into position
N40 M55	Move the milling head (C-axis) into the vertical position
N50 G54 I33	Zero point with X, Y, Z, C-table and B
...	
N100 T203 M6	Change tool in the normal spindle
N110 G0 X1000 Y2000 Z1000 C0 B0	G137 C-table active (always after M153)
N120 S3000 M3	Start the normal spindle

G136 SECOND AXES CONFIGURATION FOR FORK HEAD: ON

N130 M7	Coolant 2
N140 G7 B5=-30 L1=1	B-axis to 30 degrees
N150 G1 Z990 F3000	
...	
N370 G7	Switch off G7
N380 G174	Tool retract movement
N390 T0 M6	Normal spindle is empty
N400 M154	Move fork head into position (G137 C-table is active). C-table 90. (Zero point in C-table is 180 => real position is C270)
N410 G54 I60 C180	Set zero point C-axis
N420 G0 X1000 Y2000 Z1000	
N430 C90 A0	Position C-table and A
N440 G136	Activate C-head (fork head)
N450 T405 M6	Tool change in fork head. Only possible in G136 (C-head)
N460 G54 I60 C0.002	Set zero point C-head
N470 G0 C0 A0	C-head rotates
N480 S30000 M3	Start fork head spindle
N490 M8	Coolant 1
N500 G141 F1=5000	Activate 3D tool correction
N510 G1 Z999 F10000	
N520 X999 Y1999 Z998 I1=0 J1=0.098 K1=988.987	
...	
N10000 G40	Switch off tool correction
N10010 G174	Tool retract movement
N10020 T0 M6	Fork head spindle is empty
N10030 G137	Activate C-table. In G54 I60 is C-table 180 reactivated Position of the C-table is 90 degrees again
N10040 M153	Move the fork head out of position
N10050 M30	

General description of the second axes configuration

Format

G136

Modality

G136 and G137 are mutual modal.

Switching of the axes

G136 and G137 activate the exchange of the axes configuration.

G137 switches off the axes configuration of G136 (fork head).

Kinematic model.

The (auxiliary) axes used by G136 must be present in the kinematic model.

The machine tool needs two kinematic models for the fork head (with and without fork head)

Movement of the programmed axes

Moving to the programmed "main axes positions" in the NC-program is now done by the exchanged auxiliary axis. This is also valid for the jog buttons of the axes.

Allowed G-functions when G136 is activated:

G136 may not be programmed when G7, G8, G36, G41-G44, G64, G141, G182, G19x or G20x is active

When G136 is active, all G-functions are allowed.

Switching off G136

The function G136 is switched off with G137. G136 is not switched off by <program interrupt>, M30 or <Clear control>.

After switching on the CNC, G137 is always active. When the fork head is in position it must be therefore moved out of position or be activated by G136.

Actions

G136 and G137 refrain from all actions until the movements of the previous block are ended with <INPOD>.

Display

When G136 is active the main axes, which are exchanged by auxiliary axes become a <2> behind the relevant axes characters in front of the actual position.

During G137 the axes characters are displayed normal (without <1>).

Zero points

When an axis is exchanged by G136, resp. G137, the relevant zero point values (G52, G54, G92, G93) of these axes are also exchanged. During this the values of the switched off axes (invisible) are saved. When these axes are changed back the zero point shifts are reactivated.

The saved zero point shifts are cleared in the following cases:

- Saved value for G52 is cleared when a new pallet zero point shift or another pallet function is activated.
- Saved value for G54 Inn is cleared when a new zero point shift G54 Inn is programmed.
- Saved values for G92/G93 are cleared after programming of new G92/G93 and after M30, <cancel program> or <Clear Control>.

Note: The saved G52/G54 zero point shift values for the switched off axes are saved in the stand-by memory and are retained also after switching off the CNC.

5.60 G137 Second axes configuration for fork head: OFF

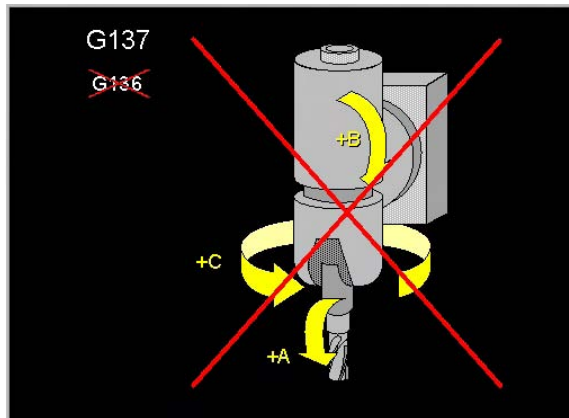
With G136 a -by the machine tool builder fixed implemented- function is deactivated (**e.g. a fork head moved into position**). The machine tool is reset to the normal axes configuration. For the possible options see the machine tool manual.

General description of the moveable fork head

The fork head activated by G136 is deactivated by G137 and the C-axis is switched back from C-head to C-table.

Format

G137



G 2nd axes config. fork head: OFF

General notes and usage

Read the description of G136 first.

Modality

G136 and G137 are mutual modal.

Exchanging the axes

G137 Switches back the axes configuration set by G136.

G137 refrains from all actions until the movements in the previous block ended with <INPOD>.

Allowed G-functions when G137 is activated:

G137 may not be programmed when G7, G8, G36, G41-G44, G64, G141, G182, G19x or G20x is active.

When G137 is active, all G-functions are allowed.

Switching off G137

The function G137 is switched off with G136. G137 is not switched off by <cancel program>, M30 or <Clear Control>.

After switching on the CNC, G137 is always active.

5.61 G141 3D-Tool correction with dynamic TCPM

Permits the correction of tool dimensions for a 3D tool path that is programmed in these points by its end point co-ordinates and normalised vectors perpendicular to the surface.

Format

To activate 3D-tool correction

G141 {R..} {R1 =..} {L2=}

To program straight-line movements

G141

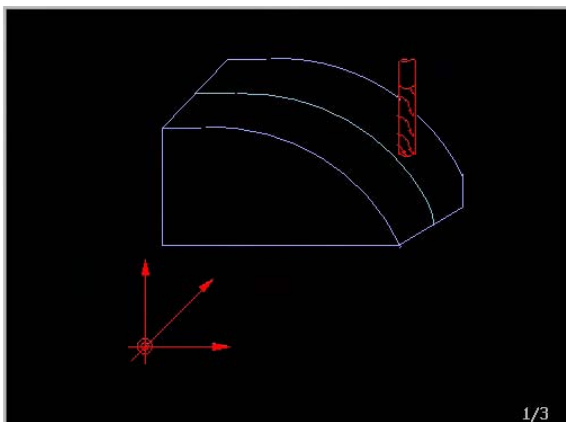
G0/G1 [end point coordinates] [I.. J.. K..]

TCPM with active kinematical model

G0/G1 [end point coordinates] {I.. J.. K..} {I1=.. J1=.. K1=..} {A, B, C} {F..}

To delete 3D-tool correction

G40



G 3D tool correction
R Nominal tool radius
L2= Rotary axes (0=shortest, 1=abs.)
R1= Nominal tool corner radius
F2= Feed limitation

With G141

R Nominal tool radius
R1= Nominal tool corner radius
L2= Circular axes (0=shortest, 1=absolute)

With G0/G1

X, Y, Z Linear end point coordinates
I, J, K Axis components of surface normal vector.
I1=, J1=, K1= (TCPM) Axis components of tool vector
A, B, C (TCPM) Circular axis components of tool vector
F Feed along the path

Associated functions

G40 and G412 to G44 for radius correction in a plane
For TCPM G8

General principles of G141

When milling a 3D surface, a given tool is moved along the surface in straight-line movements with a particular tolerance.

The calculation of the tool path on a 3D surface requires many calculations that are usually carried out by an NC programming system or a CAD system.

The calculated tool path depends on the shape of the tool, the dimensions of the tool and the tolerance to the surface.

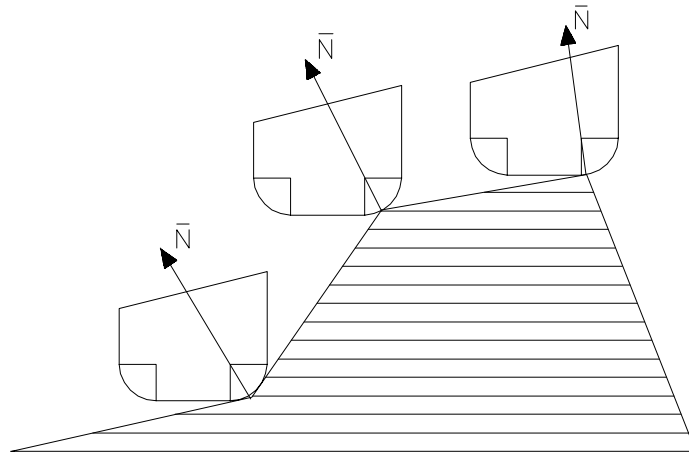
When executing the appropriate program **without G141**, the milling tool used must have the same dimensions as in the calculations, i.e. a standard milling tool must be used.

If a new tool is required while machining a 3D surface, this tool must also have the same dimensions as the **standard tool**.

If dimensional deviations are detected on the workpiece, a new calculation must be made using the programming system.

The 3D tool correction (**G141**) allows the use of tools whose dimensions differ from the dimensions of the standard milling tool. The corrections are carried out with the help of the direction vectors that are created by the programming system together with the end point co-ordinates.

In addition, the workpiece dimensions can be calculated by the programming system and the tool path by the CNC from the normalised vectors and the tool dimensions.



\bar{N} = Surface normal vector (I, J, K)

Notes and application

Radius (R, R1=)

The R.. and R1=.. values should be the same as the nominal tool dimensions used by the **programming system** for calculating the toolpath. These values are set equal to zero, if not programmed.

R defines the tool radius with which the end points of the G0/G1 blocks are calculated in the CAD system.

R1= defines the tool corner radius with which the end points of the G0/G1 blocks are calculated in the CAD system.

General principles of TCPM

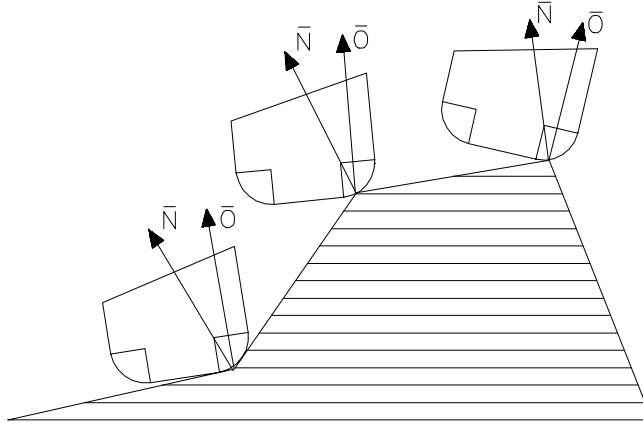
Maintaining position of tool tip when positioning swinging axes (TCPM)

(TCPM stands for "Tool Centre Point Management").

With **G141 '3D tool correction without TCMF'**, a curved (CAD) surface can be travelled taking the current tool dimensions into account. In this case, the path is described by end point co-ordinates and vectors perpendicular to the surface. The G141 function only guides the three linear axes but not the circular axes. In this way, the tool is always used in the same direction and is not guided over the workpiece surface at the optimum angle.

With **G8 'Tool orientation'** (static TCPM), the tool can be placed on the surface of the workpiece at an optimum angle. The G8 function is a feed movement and cannot be used continuously on a curved surface during a path movement.

In the case of **G141 with dynamic TCPM**, the tool is guided on a curved workpiece surface at an optimum angle. The current workpiece dimensions are taken into account. Dynamic TCPM is used for 5-axis milling. Dynamic TCPM also controls the circular axes. The tool is guided on the curved workpiece surface either vertically or at a programmed orientation.



\bar{N} = Surface normal vector (I, J, K)

\bar{O} = Tool vector (I1=, J1=, K1=) or rotary axes coordinates of the tool vector (A, B, C)

The programming format of the linear blocks within G141 is expanded to include the option of programming a tool vector. Possible combinations are surface normal vectors and/or tool vectors. If only the tool vector is used, the tool correction must be calculated in the CAD system.

G7 may be active. In this case, the surface normal vectors and the tool vectors are defined in the G7 level.

Notes and application

Addresses (R, R1=, L2=, F2=) (TCPM)

R defines the tool radius with which the end points of the G0/G1 blocks are calculated in the CAD system.

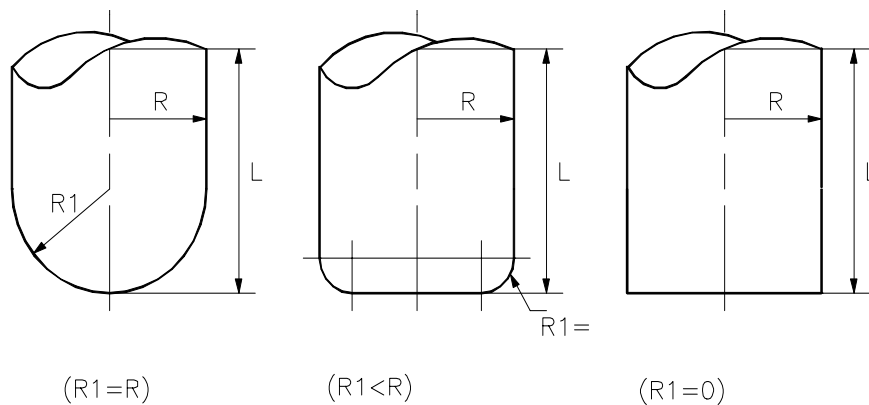
R1= defines the tool corner radius with which the end points of the G0/G1 blocks are calculated in the CAD system.

L2= 0 Circular axes travel the shortest distance (basic setting)

1 Circular axes travel to their absolute position (with circular axis programming).

F2= Feed limitation on highly curved surfaces. When radiusing an outside edge the machine may suddenly move at maximum feed. F2= limits this maximum feed. Feed override is active. F2= can only be programmed in the G141 block but it is also effective within G141 movements until the block with G40.

Possible tools



Tools used for the G141 function

Tool memory

The following dimensional details must be loaded into the tool memory to enable different types of tools to be used:

Radius milling tool	: R (tool radius), L (tool length), C (=tool radius)
Radius end milling tool	: R (tool radius), L (tool length), C (=rounding radius)
End milling tool	: R (tool radius), L (tool length), C0

If no value of C is entered, C automatically becomes 0.
The standard milling tool is thus an end milling tool.

Note: The rounding radius in the G141 block is programmed with the word R1=. The rounding radius is stored in the tool memory with the C word.

Created tool path

When the programming system creates the tool path (surface normal vector is programmed), the dimensions of the nominal tool (R.. and R1=) are programmed in the G141 block. The tool dimensions stored in the tool memory are used by the CNC to correct the tool path.

Workpiece dimensions

When the programming system creates the workpiece dimensions (surface normal vector and tool vector are programmed), the R.. and R1= words are not programmed in the G141 block. The tool dimensions stored in the tool memory are used by the CNC to calculate the tool path.

Activating G141

In the first block after G141, the milling tool travels from the current tool position to the corrected position in this block.

End point coordinates

Only absolute or incremental (X, X90, X91) Cartesian dimensional data can be used.

Up to V420, the co-ordinates in the first G141 block must be absolute and are measured from the programming zero point W.

G90/G91

The functions G90 and G91 are used for programming absolute (G90) or incremental (G91) dimensions. These functions must be alone in their own block.

Mirroring

If the mirroring function (G73 and axis co-ordinates) is active before G141 is activated, the mirrored co-ordinates are used during the 3D tool correction.

Mirroring is possible as before once G141 is activated. Mirroring is cancelled by the G73 function.

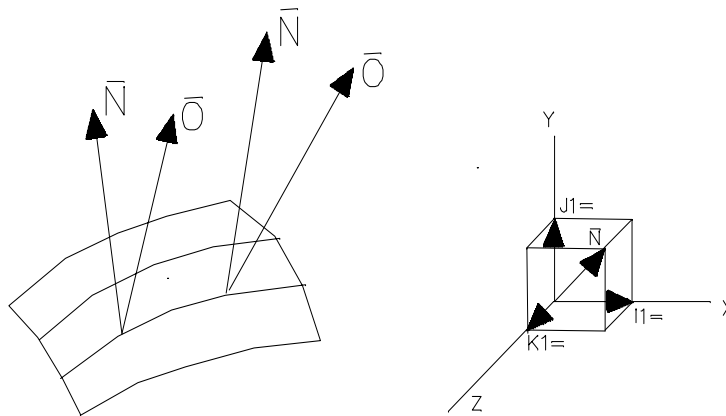
Radius correction G41...G44

After activating a G141 block, the effective radius correction programmed with G41...G44 is deleted.

Surface normal vector (I, J, K) (TCPM)

Defines the surface normal vector perpendicular to the surface.

The surface normal vector is perpendicular to the workpiece surface. The tool is positioned so that this vector always passes through the centre point of the tool corner rounding. This vector controls the positioning of the linear axes within G141. .

**Vector components**

The vector components of the axes are independent of the level selected.

If vector components are not programmed in a block, the components not programmed are set at zero.

Dimension factor

The input format of the vectors (I, J, K, I1=, J1=, K1= words) is limited to three decimal places. The surface normal and tool vectors do not, however, have to have the length 1. To increase the dimensional accuracy, the values in question can be multiplied by a dimension factor between 1 and 1000. With the factor 1000, for example, the input accuracy of the vector components is increased to six significant figures.

Back cutting

Back cutting or collisions between tool and material at points not to be machined are not detected by the CNC.

Kinematic model (TCPM)

The kinematic model is used for calculations within G141.

If no kinematic model is active (MC312 'Free machining level' = 0), G141 remains compatible with the G141 functions in older CNC versions.

Tool vector (**TCPM**)

I1=, J1=, K1= axis components of tool vector
 or
 A, B, C circular axis components of tool vector

The tool vector or the circular axis co-ordinates indicate the direction of the tool axis. The tool is turned so that it is parallel to this vector. This vector controls the positioning of the circular axes (and the associated compensation movement with linear axes) within G141.

Deleting

Function G141 is deleted by G40, M30, the program interrupt softkey or the CNC reset softkey. The milling tool stops at the last corrected position. The circular axes are not turned back automatically.

Functions to be deleted

When working with G141, functions G64, scale change (G73 A4=..), axis rotation (G92/G93 B4=..) and G182 must be deleted.

The following G functions are permitted if G141 (**TCPM**) is switched on:

Basic motions	0, 1, 7
Levels	17, 18
Program control	14, 22, 23, 29
Positioning feed	4, 25, 26, 27, 28, 94, 95, 96, 97
Radius correction	39, 40, 141
Zero points	51, 52, 53, 54, 92, 93
Geometry	72, 73
Co-ordinate measurement modes	70, 71, 90, 91
Graphics	195, 196, 197, 198, 199

If a G function that is not permissible is programmed, error message P77 'G function and Gxxx not permitted' is issued.

The following G functions are permitted if G141 (**TCPM**) is active:

Basic motions	0, 1
	Parameters of G0 and G1 are limited
	G0 without positioning logic
Program control	14, 22, 23, 29
Positioning feed	4, 25, 26, 27, 28, 94, 95, 96, 97
Radius correction	40, 141
	G40 switches G141 off
Zero points	51, 52, 53, 54, 92, 93
Geometry	72, 73
Co-ordinate measurement modes	90, 91

If a G function that is not permissible is programmed, error message P77 'G function and G141 not permitted' is issued.

Programming limitations

G functions that are not listed above may not be used.
 Point definitions (P) and E parameters may not be used.
 No tool change may be made after activating G141.

Notes and application for TCPM**Risk of collision**

When G141 is switched on, compensation movements similar to those in G8 may occur.

In the case of the switch-on movement, the tool tip must not be resting on the surface of the workpiece and should be programmed with a distance from the material at least equal to the tool diameter.

Remark: If G141 is switched off via G40, M30 or program cancel, there is no compensation movement and the circular axes remain in their last positions.

When approaching the contour, it may happen that the table with the workpiece is turned through 180 degrees to achieve the programmed tool direction. **ATTENTION! RISK OF COLLISION!**

Undercutting

If the tool direction changes within a G1 block, this tool direction change is carried out interpolating with the movement to the end point. In doing this, the path between the start and end points is corrected for undercutting.

Undercutting is not detected during block transitions. This undercutting should be corrected by inserting a block without an end point and with only one change of the tool vector by the CAD system. In this case, the tool turns about the tool contact point until the new tool direction is reached.

Display

When G141 is active, a yellow icon is displayed behind the tool number and the programmed G141 tool vectors (I1, J1, K1) can be seen in the machining status (on the G7/G8 positions).

Remark: If G7 and G141 are active at the same time, the G7 angle or vector can be seen.

A small 'p' at the bottom right, near the 'axis letters', shows whether the position of the tool contact point or the position is in machine co-ordinates. The display changes with the same softkey as with G7.

Feedrate

The programmed feedrate applies to the contact point between the surface and the tool. The tool head may make other movements.

Error messages**P341 Tool vector incorrect**

The tool vector (I1=, J1=, K1=) is incorrect. This error message is generated if all the components of the vector are zero.

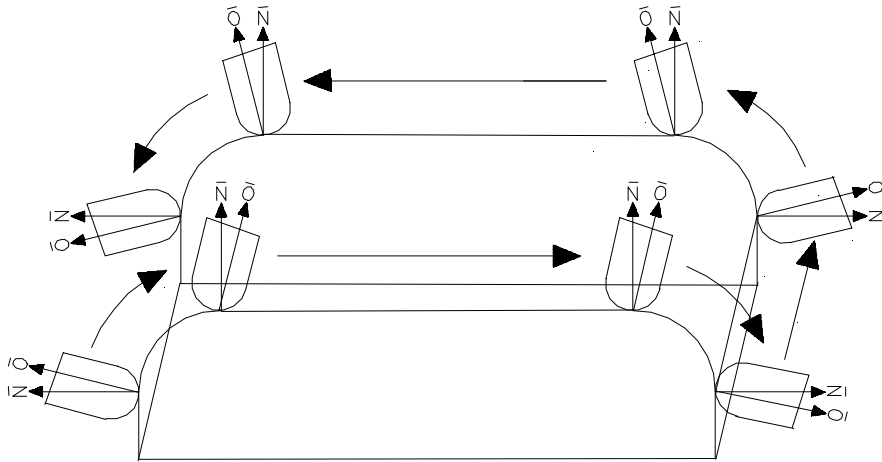
P342 Surface normal vector incorrect

The surface normal vector (I, J, K) is incorrect. This error message is generated if all the components of the vector are zero.

Example**Example 1** G141 and TCFM

Tool vector with (I1=, J1=, K1=)

This program is independent of the machine.



N113 (square material with top rounding (R4) and swung tool (5 degrees))

N1 G17

N2 T6 M67 (10 round spherical milling tool: T6 R5 C5 in tool table)

N3 G54 I10

N4 G0 X0 Y0 Z0 B0 C0 S6000 M3

N5 F50 E1=0

N6 G141 R0 R1=0 L2=0 (all basic settings, do not need to be programmed)

N7 (R in CAD System is 0 mm)

N8 (R1 in CAD System is 0 mm)

N9 (L2=0 circular axes move shortest distance)

N10

N11 G0 X-1 Y=E1 Z0 I1=-1 K1=0

N12 (generated in CAD System)

N13 (front left arc)

N14 G1 X=0 Y=E1 Z=-4 I1=-0.996194698 K1=0.087155743

N15 G1 X=0.000609219 Z=-3.930190374 I1=-0.994521895 K1=0.104528463

N16 G1 X=0.002436692 Z=-3.860402013 I1=-0.992546152 K1=0.121869343

N17 G1 X=0.005481861 Z=-3.790656175 I1=-0.990268069 K1=0.139173101

N... (Each degree a point)

N100 G1 X=3.790656175 Z=-0.005481861 I1=0.034899497 K1=0.999390827

N101 G1 X=3.860402013 Z=-0.002436692 I1=0.052335956 K1=0.998629535

N102 G1 X=3.930190374 Z=-0.000609219 I1=0.069756474 K1=0.99756405

N103 G1 X=4 Z=0 I1=0.087155743 K1=0.996194698

N104 (front right arc)

N105 G1 X=36 Z=0 I1=0.087155743 K1=0.996194698

N106 G1 X=36.06980963 Z=-0.000609219 I1=0.104528463 K1=0.994521895

N107 G1 X=36.13959799 Z=-0.002436692 I1=0.121869343 K1=0.992546152

N...

N194 G1 X=39.99756331 Z=-3.860402013 I1=0.998629535 K1=-0.052335956

N195 G1 X=39.99939078 Z=-3.930190374 I1=0.99756405 K1=-0.069756474

N196 G1 X=40 Z=-4 I1=0.996194698 K1=-0.087155743

N197 G40
 N1971 (back right arc)
 N1972 (move up to next cut)
 N1973 G174 I100 (tool withdrawal)
 N1974 G0 B0 C0 (rotate circular tables to original coordinates system)
 N198 E1=E1+0.25
 N1981 G1 Y=E1 (movement in normal X, Y, Z coordinates system)
 N1982 G141

OR without deactivation G141

N197 (back right arc)
 N198 E1=E1+0.25 (move up to next cut)

N199 G1 X=40 Y=E1 Z=-4 I1=0.996194698 K1=0.087155743
 N200 G1 X=39.99939078 Z=-3.930190374 I1=0.994521895 K1=0.104528463
 N201 G1 X=39.99756331 Z=-3.860402013 I1=0.992546152 K1=0.121869343

N...

N287 G1 X=36.13959799 Z=-0.002436692 I1=-0.052335956 K1=0.998629535
 N288 G1 X=36.06980963 Z=-0.000609219 I1=-0.069756474 K1=0.99756405
 N289 G1 X=36 Z=0 I1=-0.087155743 K1=0.996194698
 N290 (back left arc)
 N291 G1 X=4 Z=0 I1=-0.087155743 K1=0.996194698
 N292 G1 X=3.930190374 Z=-0.000609219 I1=-0.104528463 K1=0.994521895
 N293 G1 X=3.860402013 Z=-0.002436692 I1=-0.121869343 K1=0.992546152

N...

N379 G1 X=0.002436692 Z=-3.860402013 I1=-0.998629535 K1=-0.052335956
 N380 G1 X=0.000609219 Z=-3.930190374 I1=-0.99756405 K1=-0.069756474
 N381 G1 X=0 Z=-4 I1=-0.996194698 K1=-0.087155743
 N382 E1=E1+0.25

N383 G14 N1=10 N2=389 J40

N384 G40
 N385 G174 I100 (tool withdrawal movement)
 N386 G0 B0 C0 (rotate circular tables to original coordinates system)
 N387 M30

Example 2 G141 and TCPM

Identical workpiece
 Tool vector with (A, B, C)
 This program is machine dependent.

This program is for a machine with on the table a B-Axes under 45°, with upon a C-axes.

N114 (Rectangle block with rounding on top (R4) and tilting tool position (5 degrees))
 N1 G17
 N2 T6 M67 (Ball cutter round 10: In tool table T6 R5 C5)
 N3 G54 I10
 N4 G0 X0 Y0 Z0 B0 C0 S6000 M3
 N5 F50 E1=0

N6 G141 R1=0 L2=0 (all default, so not necessary to program)
 N7 (R in CAD System is 0 mm)
 N8 (R1 in CAD System is 0 mm)
 N9 (L2=0 Rotary axes moves shortest way)
 N10
 N11 G0 X-1 Y=E1 Z0 B180 C-90
 N12 (generated in CAD System)
 N13 (front arc left)
 N14 G1 X=0 Y=E1 Z=-4 B145.658 C-113.605
 N15 G1 X=0.000609219 Z=-3.930190374 B142.274 C-115.789
 N16 G1 X=0.002436692 Z=-3.860402013 B139.136 C-117.782
 N17 G1 X=0.005481861 Z=-3.790656175 B136.191 C-119.624

N... (Each degree a point)

N100 G1 X=3.790656175 Z=-0.005481861 B2.829 C1
 N101 G1 X=3.860402013 Z=-0.002436692 B4.243 C1.501
 N102 G1 X=3.930190374 Z=-0.000609219 B5.658 C2.001
 N103 G1 X=4 Z=0 B7.073 C2.502
 N104 (front arc right)
 N105 G1 X=36 Z=0 B7.073 C2.502
 N106 G1 X=36.06980963 Z=-0.000609219 B8.489 C3.004
 N107 G1 X=36.13959799 Z=-0.002436692 B9.906 C3.507

N...

N194 G1 X=39.99756331 Z=-3.860402013 B206.449 C108.384
 N195 G1 X=39.99939078 Z=-3.930190374 B210.629 C111.170
 N196 G1 X=40 Z=-4 B214.342 C113.605
 N197 (back arc right)
 N198 E1=E1+0.25 (now translation)
 N199 G1 X=40 Y=E1 Z=-4 B145.658 C66.395
 N200 G1 X=39.99939078 Z=-3.930190374 B142.274 C64.211
 N201 G1 X=39.99756331 Z=-3.860402013 B139.136 C62.218

N...

N287 G1 X=36.13959799 Z=-0.002436692 B4.243 C-178.499
 N288 G1 X=36.06980963 Z=-0.000609219 B5.658 C-177.999
 N289 G1 X=36 Z=0 B7.073 C-177.498
 N290 (back arc left)
 N291 G1 X=4 Z=0 B7.073 C-177.498
 N292 G1 X=3.930190374 Z=-0.000609219 B8.489 C-176.996
 N293 G1 X=3.860402013 Z=-0.002436692 B9.906 C-176.493

N...

N379 G1 X=0.002436692 Z=-3.860402013 B206.449 C-71.616
 N380 G1 X=0.000609219 Z=-3.930190374 B210.629 C-68.830
 N381 G1 X=0 Z=-4 B214.342 C-66.395
 N382 E1=E1+0.25

N383 G14 N1=14 N2=382 J40

N384 G40

N385 G174 L100 (Retract tool)

N386 G0 B0 C0 (turn rotary tables to original coordinates system)

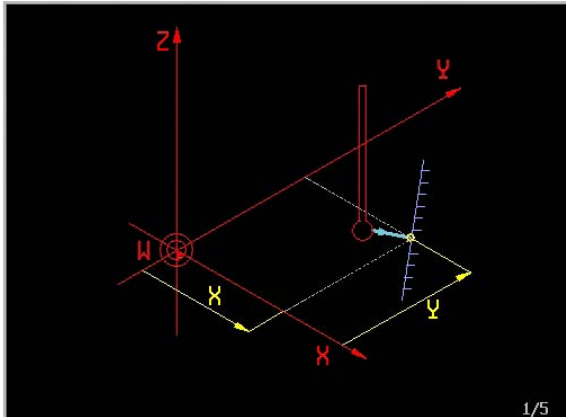
N387 M30

5.62 G145 Linear measuring movement

To execute a free programmable linear measuring movement to determine axis positions in measuring cycle macros.

Format

G145 [point to be measured] [(axis address) 7=...] {S7=...} E... {F2=...} {K...} {I3=...} {I4=...}
G0 [axis coordinates]



G Linear measuring movement
X Endpoint coordinate
Y Endpoint coordinate
Z Endpoint coordinate
B Endpoint angle
C Endpoint angle
K 0=tool correction on, 1=off
E E-parameter for measuring status
B1= Angle
B2= Polar angle
X7= E-par. for measured value in X
Y7= E-par. for measured value in Y
Z7= E-par. for measured value in Z
B7= E-par. for measured value in B
C7= E-par. for measured value in C

?90= Endpoint abs. (X,Y,Z..)
?91= Endpoint incr. (X,Y,Z..)
I3= Status control (0=on, 1=off)
I4= Air supply (0=off, 1=on)
L1= Path length
L2= Polar length
P1= Point definition number
F2= Measuring feed
S7= E-par. for measured value in S

Notes and usage

Associated Functions

G148, G149, G150, G45, G46
M24, M26, M27, M28, M29

Measuring conditions (L)

Measuring is always done by contact.

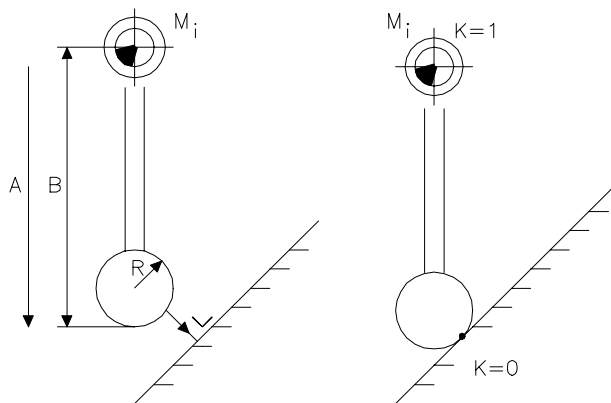
Storage measured value [(Axis address)7=]

This word states the E-parameter number which is to contain the measured axis position; e.g. X7=2, states that X-axis measured value must be stored in parameter E2; X7=E1 (E1=5) means measured value is stored in E5.

For example: S7=... spindle angle will be store in E-parameter 5.

Note: Before S7=... must be programmed an oriented spindle stop (M19). Otherwise has the spindle

Compensating for probe dimensions (K)



A: Tool axis
B: Tool length

- K0: Tool correction on. Measured positions are corrected for the tool length and radius. Measured positions of rotary axes are not corrected for tool data.
- K1: Tool correction off. Measured positions are not corrected. If K is not programmed, K0 is automatically activated.

The following assumptions are used when the measured positions are corrected for the probe dimensions.

- The probe is parallel to the tool axis.
- The probe is perfectly spherical.
- The probe movement is perpendicular to the surface being measured.

Measuring probes status (E)

The measuring probe can be in one of three states after completion of a G145 block.

The assigned E-parameter can therefore have one of three states.

- E... = 0 the programmed position has been reached, but no measuring point has been found. The assigned E-parameters, which contain measuring values, remain unaltered.
- E... = 1 during the measuring movement a measuring point has been found. The measured position of the axes has been entered into the E-parameters.

Status control (I3= 0=on, 1=off) (Status of the turning aside of the probe)

The status control of the measuring probe inside the G145 can be disconnecting for certain devices (laser). The default is zero.

Measuring feedrate (F2=)

If F2= is not programmed a default value stored in a machine constant (MC843) is used automatically.

Note: If a function for spindle direction (M03 or M04) is entered, this function will be suppressed and an error code output.

The function G145 is not permitted, when G182 is active.

Air supply (I4=) (0=no 1=yes)

The air supply duration before the measuring is stored in Machine constant (MC842). (Default is 0)

Block search

During BLOCK SEARCH the measuring movement is simulated. The E-parameters, in which the measured coordinates were to be loaded, remain unaltered. The signals from the measuring probe are ignored.

Demo

During DEMO a movement is executed towards the programmed position. The programmed coordinates are loaded into the E-parameters. The signals from the measuring probe are ignored.

Testrun

During TESTRUN the measuring movement is executed with the test feedrate (a Machine Constant (MC741)) or simulated if TESTRUN is executed without movements. The programmed coordinates are loaded into the E-parameters. When the probe is triggered during a movement, the movement is aborted and a collision error generated.

Graphics

In the GRAPHICS modes the measuring movements are simulated. The programmed coordinates are loaded into the E-parameters. The signals from the measuring probe are ignored.

Note: In all the mentioned operation modes, the E-parameter for the measuring probe status gets the value 2. By checking this parameter in the measuring macros it is possible to avoid using parameters, which do not contain measured values.

Intervention

With INTERVENTION the G145-movement is treated as a G1-movement. The status of the probe should not be changed between the start point of the measuring movement and the point of interrupt. If the status was changed, an error message is displayed. An error message is also displayed, if the probe is triggered during the repositioning.

Examples**Example 1** Measuring tool length with a measuring box

Two macros and a program are given for measuring the tool length with the aid of a measuring box.

In the first macro (N14501) the trigger point of the box is determined.

In the second macro (N14502) the actual length measurement is executed.

In the program (N14503) parameters are set and both macros called.

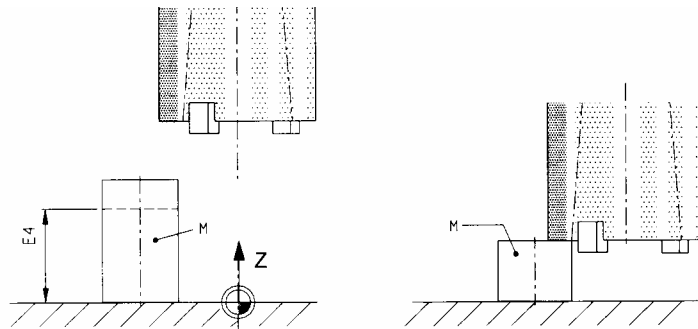
Note

1. Refer to the machine tool builder's documentation to see if a measuring box can be used and which M-functions have to be used for switching on and off the measuring box in a particular installation.
2. The program and macros are given to show the possibilities of the measuring cycles and the E-parameters. Further update might be necessary to adapt the macros to the specific requirements of the user.

Used parameters

E0:	E-Parameter for jump function
E1:	X-coordinate of the measuring box
E2:	Y-coordinate of the spindle reference point
E3:	Z-coordinate of the measuring box
E4:	Y-coordinate of the trigger point of the box
E5	=0: Trigger point is not determined
	=1: Trigger point is already determined
E7	=0: No error found in macro N14501
	=1: An error found in macro N14501
E8:	Tool number or Tool identification number
E10:	Measured Y-coordinate

Macro for determination the trigger point of the measuring box.



M = Measuring box

E4 = Trigger point position

N14501 (Macro trigger position of the measuring box)

N1 T0 M6

Unload the spindle

N2 M24

Activate with M24 the measuring

N3 G0 X0 Y0 Z150

Move the spindle nose above the measuring box

N4 G145 Z20 E7 F2=2000

Measuring movement in the Z-axis (=tool axis). A large Z-position is used so that it is sure that the box is reached. Store measuring position in E4.

N5 G0 Z150

Retract the spindle

N6 G29 E0=E7=1 E0 N=9

Check to see if the box is reached. (E7=1!). Jump to the end of the macro

N7 M0

Program stop

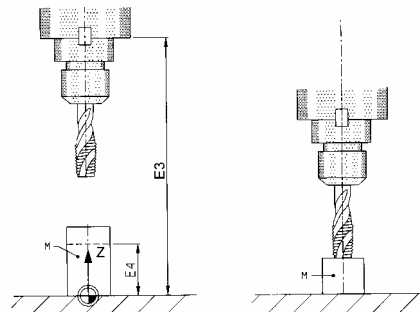
N8 (NO TRIGGER POINT FOUND)

Message, that the trigger point is not found.

N9 ...

End of macro.

Macro for measuring the tool length.



M = Measuring box

E4 = Trigger point position

N14502 (Macro for measuring the tool length)

N1 T=E8 M6

Load the tool to be measured

N2 G0 X=E1 Y=E2 Z=E3

Move the tool above the measuring box

N3 G145 Z=(E4-20) E7 F2=2000

Measuring movement in the Z-axis (=tool axis). A Z-position past the trigger point is used so that it is sure that the box is reached. Measure at contact.

N4 G29 E0=E7 <1 E0 N=8

Check to see if the box is reached. (E7=1!)

N5 G0 Z=E3

Retract the tool

N6 G150 T=E8 L1=E10-E4

Update length of the actual tool in the tool memory

N7 G29 E0 E0=1 N=10

Jump to the end of the macro

N8 M0

Program stop

N9 (length measurement unsuccessful)

Message that tool length is not measured

N10 ...

End of the macro

Program for measuring tool length with a measuring box

The first time the program is used, the user has:

- to move the spindle nose above the measuring box
- to set the zero point with PRESET AXES
- to enter the parameters E5=0 and E8

If more tools have to be measured, parameter E5=1 has to be set and parameter E8 (the tool number) to be entered.

N14503 E8=.. (TOOL NUMBER)

N1 E5=0 (=0 determine trigger point =1 no trigger point) Set the parameters E5 and E8

N2 G17

Set the plane of operation to be the XY-plane

N3 G54

Set the zero point found with PRESET AXES

N4 G29 E5 K0 N=8

Check to see if the trigger point has to be determined (E5=0) or can be ignored.

N5 G22 N=14501

Call macro N=14501 to determine the trigger point of the measuring box

N6 G29 E0 E0=E7<1 N=9

Check to see if an error was found in macro N14501 (E7=1); in that case ignore the next macro call.

N7 E1=0 E2=0 E3=350

Set the parameters containing the coordinates of the box

N8 G22 N=14502

Call macro N14502 to measure a tool length

N9 M28

Switch off with M28 the measuring box

N10 T0 M6

Unload the spindle

N11 G53

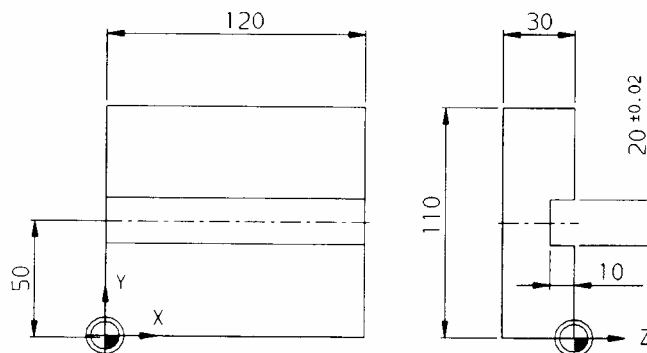
Restore the zero point

N12 M30

End of the program

Example 2 Milling and measuring a groove

Milling a groove followed by measuring the width of the groove. If the width is too small, the radius of the mill is changed and the groove is finished again.



N14504 (milling and measuring a groove)

N1 G17

Define the plane of operation

N2 G54

Activate a stored zero offset

N3 E15=20.02 (maximum width of the groove) Set the maximum width of the groove

N4 E16=19.98 (minimum width)

Set the minimum width of the groove

N5 E3=(E15+E16):2

Calculate the width of the groove with the average tolerance

N6 T1 M6 (MILL Diameter 18 mm)

The mill of 18 mm diameter is loaded as tool 1

N7 G0 X-25 Y50 Z-10 F400 S1000 M3

Start the spindle and move tool to start position

N8 G1 X140

Mill through the centre of the groove

N9 G43

Mill the sides of the groove

N10 G1 Y60

N11 G41

N12 X-25

N13 Y40

N14 X140	
N15 G40	Cancel radius compensation
N16 Y50	Move the tool to the middle of the groove
N17 G0 Z50 M5	Retract the tool and stop the spindle
N18 G149 T0 E30	Pick up the number of the actual tool
N19 T30 M6 (TOUCH TRIGGER PROBE)	Load the touch trigger probe
N20 D207 M19	Set the probe in an oriented position. It depends on the machine tool if this setting is necessary.
N21 M27	Activate the probe
N22 X60 Y50 Z-8	Move the probe to the middle of the groove and at depth
N23 M29	Activate the air pressure. The code of the M-function depends on the machine tool
N24 G145 Y65 E10 Y7=1 F2=500	Measure the upper side of the groove. Store the measured value in the Y-axis at E1. Store the status of the measuring probe at E10
N25 G0 Y50	Move the tool back to the middle of the groove
N26 G29 E11=E10=0 E11 N=29	Check to see if a measurement was executed E10=0 no measurement. Jump to N29 to switch off the probe and to display an error message.
N27 M29	Activate the air pressure.
N28 G145 Y35 E10 Y7=2 F2=500	Measure the lower side of the groove. Store the measured value in the Y-axis at E2 Store the status of the measuring probe at E10
N29 G0 Y50	Move the tool back to the middle of the groove
N30 M28	Switch off the probe
N31 G29 E11=E10=0 E11 N=41	Check to see if a measurement was executed E10=0 No measurement. Jump to N41 to display an error message.
N32 E5=E1-E2	Calculate the actual width of the groove from the measured Y-positions
N33 E6=(E5-E3):2	Calculate the difference between the programmed width and the measured width. The difference is related to the tool radius.
N34 G29 E20=E5>E15 E20 N=43	Check to see if the measured tool width (E5) is greater than the maximum allowed width (E15). If greater jump to N43 to display an error.
N35 G29 E20=E5>E16 E20 N=45	Maximum width is not exceeded. Check to see if the measured width (E5) is greater than the minimum allowed width. If greater the groove is finished. So jump to the end of the program (N44).
N36 G149 T=E30 R1=4	The measured width is less than the minimum value. In this case the tool radius is changed in the tool memory and the sides of the groove milled again with the new radius value from the tool memory. Read the tool radius from the tool memory and store its value in parameter E4.
N37 G150 T=E30 R1=E4+E6	Store the recalculated tool radius in the tool memory
N38 S1000 T1 M6 (MILL Diameter 18 mm)	The mill is loaded again
N39 G0 X140 Y50 Z-10 B0 F400 M3	Start the spindle and move tool to start position
N40 G29 E20 E20=1 N=43	If the milling is finished, jump to the end of the program.
N41 M0	Program stop
N42 (probe not triggered, no measurement executed)	Displaying an error text
N43 G29 E20 E20=1 N=43	If the program continues, a jump to the end of the program is executed
N44 M0	Program stop
N45 (GROOVE WIDTH TOO BIG)	Displaying an error text
N46 M30	End of program

Example 3 Alignment of workpiece mounted on a rotary table

Only two points in X or Y have to be measured to be able to adjust a workpiece mounted on the rotary table rotating around the Z axis.

The angle between workpiece and X axis is calculated automatically and may be used to rotate the table for positioning the workpiece parallel to the X axis.

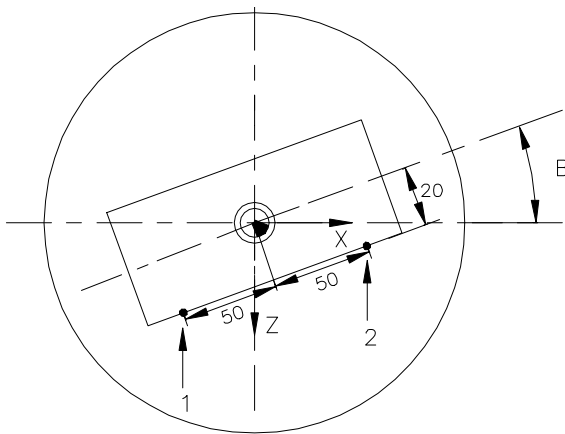
If the workpiece is inclined towards the X axis at the beginning, this angle may be programmed with the C word. If C has not been programmed, C0 is used automatically instead of C.

Note:

This function can only be used when:

1. the rotary table is located in the XY plane, the workpiece is rotated around the Z axis (C axis) and the measuring probe is in Z direction.
2. the measurements are carried out in Y direction.

A part is mounted on a rotary table and should be aligned parallel to the X-axis. With a touch trigger probe two points on the part are measured and then the table is rotated over the calculated angle.



N50003

N1 G17

N2 G54

N3 T1 M6

N4 G0 X-50 Y-30 Z100 C0

N5 G1 Z0

N6 M27

N7 G145 X-50 Y-20 X7=11 Y7=12

N8 G0 Y-30

N9 G0 X50

N10 G145 X50 Y-20 X7=21 Y7=22

N11 G0 Y-30

N12 M28

N13 G0 Z100

N14 E30=(E12-E22)

N15 E31=(E21-E11)

N16 E32=(E30:E31)

N17 E33=atan(E32)

N18 G150 C7=E33 N1=54

N19 G54

N20 G0 C0

N21 M30

Set the plane of operation

Set the zero point

Load the touch trigger probe

Positioning first measuring point

Going to depth

Activate probe

Measure point 1 in direction Y (X in E11, Y in E12)

Retract

Positioning second measuring point

Measure point 2 in direction Y (X in E21, Y in E22)

Retract

Deactivate probe

Retract Tool

Calculate the difference (E30) in Y direction

Calculate the difference (E31) in X direction

Calculate Quotient (E32)

Calculate ARCTAN (E33)

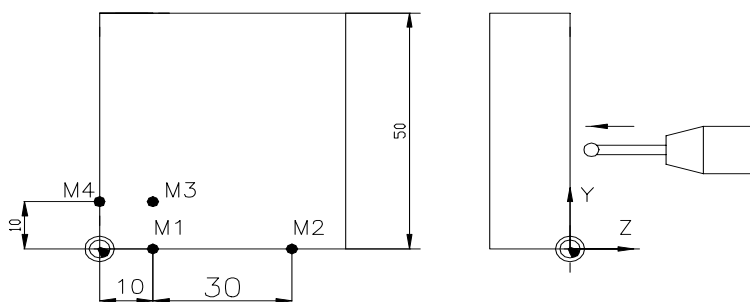
A angle correction is made in the zero point of the C-Axis

Set the zero point

Rotate table back to C0

Program end N19 :

Example 4 Determining the zero point



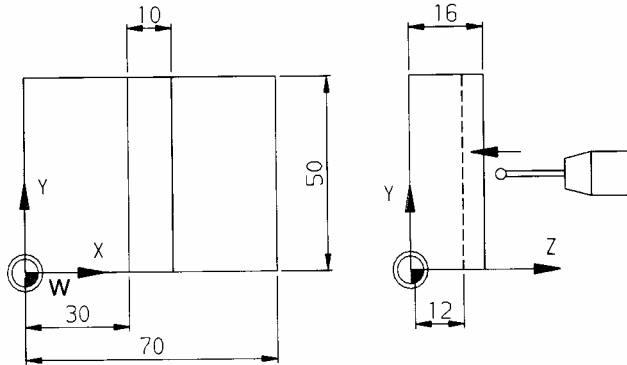
The probe is standing in the Z-axis. The part is mounted on a table rotating around the Z-axis. Five points of the part (M1 to M5) are measured. M1 and M2 for covering the angular displacement; M3, M3 and M5 for measuring the positions of the axes.

The section of the part program for determination the zero point could be:

N50004	
N1 G54	Set the zero point
N2 G17	Set the plane of operation to be XZ-plane
N3 T1 M6 (probe)	Load the touch trigger probe
N4 G0 X10 Y-10 Z70 C0 F1000	Move to programmed position
N5 G1 Z-5	move to depth in hole 1
N6 M27	Activate probe
N7 G145 X10 Y0 X7=11 Y7=12	Measure point M1 (X in E11, Y in E12)
N8 G0 Y-10	Retract probe to avoid collision
N9 G0 X50	Move to position for measure point 2
N10 G145 X50 Y0 X7=21 Y7=22	Measure point M2 (X in E21, Y in E22)
N11 G0 Y-10	Retract probe to avoid collision
N12 G0 Z70	Retract probe to avoid collision
N13 E30=(E12-E22)	Calculate the difference (E30) in Y direction
	N14 E31=(E21-E11) Calculate the difference (E31) in X direction
N15 E32=(E30:E31)	Calculate quotient (E32)
N16 E33=atan(E32)	Calculate ARCTAN (E33)
N17 G150 C7=E33 N1=54	A angle correction is made in the zero point of the C-Axis
N18 G54	Set the zero point
N19 G0 C0	Rotate table back to C0
N20 G0 X10 Y10 Z10	Move to measure point 3
N21 G145 X10 Y10 Z0 Z7=3	Measure point M3, to determine the position in the Tool axis(Z in E3)
N22 G0 Z10	Retract probe to avoid collision
N23 G0 X-10 Y10	Move to measure point 4
N24 G1 Z-5	Move to depth
N25 G145 X0 Y10 Z-5 X7=1	Measure point M4, to determine the position in the X axis(X in E3)
N26 G0 X-10	Retract probe to avoid collision
N27 G0 Z10	Move to measure point 1
N28 G0 X10 Y-10	Move to depth
N29 G1 Z-5	Measure point M1, to determine the position in the Y axis(Y in E3)
N30 G145 X10 Y0 Z-5 Y7=2	Retract probe to avoid collision
N31 G0 Y-10	

N32 G0 Z50
 N33 G150 X7=E1 Y7=E2 Z7=E3 N1=54 Update of the zero offset values in the X-, Y- and Z-Axis
 N34 G54 Set the updated zero point
 N35 M28 Deactivate probe

Example 5 Correcting the length of a tool



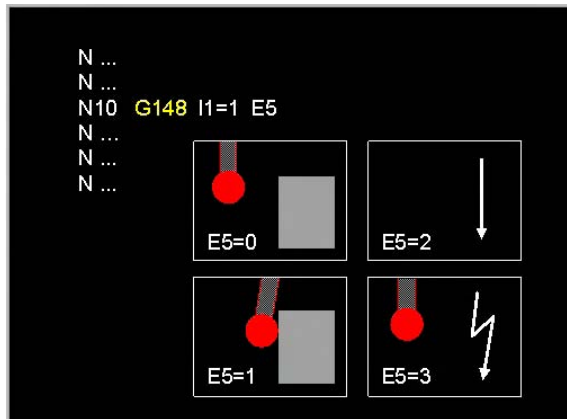
N90005	
N1 G17	Set the plane of operation to be the XY-plane
N2 T1 M6 (Mill R5)	Load the mill of 10 mm diameter
N3 G0 X35 Y60 Z12 S1000 M3	Start the spindle and move the mill to the start point
N4 G1 Y-10 F200	Mill the groove
N5 G0 Z200 M5	Retract the tool and stop the spindle
N6 T2 M6 (Prober)	Load the probe
N7 G0 X35 Y25 Z20	Move to start point
N8 M27	Activate probe
N9 G145 X35 Y25 Z12 Z7=1	Measure the tool axis in negative direction (Z in E1)
N10 G149 T1 L1=2	Store the tool length T1 in E2
N11 G150 T1 L1=E2-E1+12	The tool length is dependent of the calculated difference in the Z-Axis.
N12 M28	Deactivate probe
N13 Z200 M30	Retract probe and end program.

5.63 G148 Reading measure probe status

To read the probe status in measuring cycle macros.

Format

G148 {I1=...} E...



G Read measure probe status
E E-parameter for probe status
I1= Status group (1-3)

Notes and usage

Associated Functions

G145, G149, G150

Probe status

I1=1 or not programmed (default).

The E-parameter can have one of four values:

- | | |
|----------|---|
| E... = 0 | Probe not deflected. |
| E... = 1 | Probe deflected. |
| E... = 2 | Block Search, testrun or the Demo mode is active. |
| E... = 3 | A probe error is active; no measurements can be made. |

The priority for probe status codes is:

- 1: code 2 (active mode)
- 2: code 3 (probe error)
- 3: code 0 or 1 (probe contact)

- | | | |
|------|----------|---|
| I1=2 | E... = 0 | during the measurement no measuring point is determinate. |
| | E... = 1 | during the measurement a measuring point is determinate. |
| I1=3 | E... = 0 | information of IPLC: probe/laser not connect |
| | E... = 1 | information of IPLC: probe/laser connect |

Interrupt

The G148 function cannot be stopped by an interrupt command.

Example

N110 G148 E27

N115 G29 E91=E27=2 E91 N=

Store probe status in E-parameter number 27.

Jump to block N300 if the program is executed in Block Search, Testrun or Demo. In this way e.g. calculation with parameters, which are not loaded while no measurement was executed, can be avoided.

Note: The function G148 is not permitted when G182 is active.

5.64 G149 Reading tool data or zero offset values

To read tool data or zero offset values and store them in specified E-parameters within measuring cycle macros.

Format Tool data

To read active tool number:

G149 T0 E...

To read tool dimensions:

G149 T... {T2=...} {L1=...} {R1=...} {M1=...}

To read tool status:

G149 T... E...

Format Zero offsets

To read active zero offset G-functions:

G149 N1 =0/1 E...

To read stored pallet offsets:

G149 N1=52 [(axis address)7=...] {(axis address)7=...}

To read stored zero offsets:

With standard zero offsets or MC84=0:

G149 N1=54...59 [(axis address)7=...] {(axis address)7=...}

With MC84>0 zero offsets extends:

G149 N1=54.[nr] [(axis address)7=...] {(axis address)7=...}{B47=...}

In G54.[nr] the number [nr] must be given in two digits (G54.01 and G54.10).

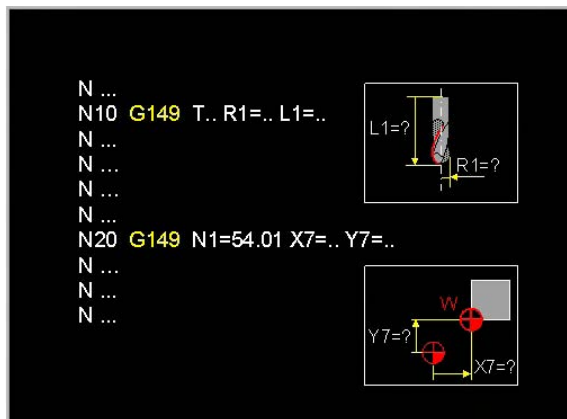
To read programmable zero offset:

G149 N1 =93 [(axis address)7=...] {(axis address)7=...}

Format: Actual position

To read the actual position of X, Y or Z:

G149 [(axis address)7=...] {(axis address)7=...}



G Read tool or zero offset values
 T Tool number
 E E-parameter
 N1= Zero offset shift
 B47= E-parameter for rotation in B4=
 X7= E-par. for offset/position in X
 Y7= E-par. for offset/position in Y
 Z7= E-par. for offset/position in Z
 B7= E-par. for offset/position in B
 C7= E-par. for offset/position in C
 L1= E-parameter for toollength
 R1= E-parameter for tool radius
 T2= Tool offset index
 M1= E-parameter for toollife

Tool data

T Tool number
 T2= Tool offset index
 E E-parameter
 L1= E-parameter for toollength
 R1= E-parameter for tool radius
 M1= E-parameter for toollife

Zero offsets

N1= Zero offset shift
 X7= E-parameter for offset /position in X
 Y7= E-parameter for offset /position in Y
 Z7= E-parameter for offset /position in Z

A7=	E-parameter for offset /position in A
B7=	E-parameter for offset /position in B
C7=	E-parameter for offset /position in C
B47=	E-parameter for rotation in B4=

Notes and usage

Associated Functions

G145, G148, G150

Tool number (T)

Number of tool for which tool data must be read. If the FMS-tool memory (Flexible Manufacturing System) is in use, the complete number including the spare tool index has to be written.

Tool data

The tool radius (R1=..), toollength (L1=..) and the remaining toollife (M1=..) can be read.

Tool offset index (T2=)

A tool-offset index 0,1 or 2 can be specified. Default is T2=0.

When T2=0

Tool radius = radius (R) + radius oversize (R4=)

Toollength = length (L) + length oversize (L4=)

Better is to use G321.

Tool status (E)

The tool status from the tool memory will be loaded in the indicated E-parameter. The tool status can have the following values.

E... = 1	the tool is enabled and measured.
E... = 0	the tool is enabled but not measured
E... = -1	the tool is disabled
E... = -2	tool time is expired
E... = -4	tool breakage error
E... = -8	the tool cutting force is exceeded
E... = -16	the tool time < T3 = programmed

A combination of error messages is also possible:
E... = -13 means: error message -8 and -4 and -2 and 1.

Zero offset number (N1=)

The number of the zero offset which data has to be read. 'N1=' can have a value from 51 to 59, 54.[nr] or 92/93. G92 gives the same result as G93 (absolute).

Zero offset group (N1=)

The zero offset group of which the active G-function has to be read.

N1= can have the value 0 or 1.

N1=0 If G52 is active, the E-parameter is given the value 52. If G52 is not active, the E-parameter is given the value 51.

N1=1 The E-parameter is given the value of the active offset G54 - G59. IF a G54-G59 type offset is not active, the E-parameter is given the value 53.

Reading actual axes position values (X7, Y7, Z7)

The Axes position values can be read out in E-Parameter. X7=20 means: E20 is filled with the actual axes position values. (See also G326).

Reading addresses without value:

If addresses are read from the tool memory when they are not entered previously, a value of zero will be returned.

Interrupt

The function cannot be stopped by an interrupt command.

Note: The function G149 is not permitted when G182 is active.
The tool data of T0 cannot be read. If T0 is used, the relevant E-parameters are not loaded.
No error message is given to this effect.

Examples

Example 1 to read the active tool number.

N100 G149 T0 E1 E1 contains the number of the active tool

Example 2 to read the active tool dimensions.

N100 G149 T=E1 L1=5 R1=6 M1=7 Read the tool dimensions of tool T=E1
E5 contains the tool length (E5 = length (L) + length oversize (L4=))
E6 contains the tool radius (E6 = radius (R) + radius oversize (R4=))
Better is to use G321.
E7 contains the rest tool life time

Example 3 to read the active tool dimensions.

N100 G149 T12 L1=5 R1=6 Read the tool dimensions of tool T12
E5 contains the tool length (E5 = length + length oversize (L4=))
E6 contains the tool radius (E6 = radius (R) + radius oversize (R4=))
Better is to use G321.

Example 4 to read the active zero offset function

N100 G149 N1=0 E2 E2 contains the active preset function (51 or 52)
N110 G149 N1=1 E3 E3 contains the active zero offset function (53 to 59) or G54.[nr]

Example 5 to read a stored zero offset.

N100 G149 N1=54 X7=1 Z7=2
or
N100 G149 N1=54.[nr] X7=1 Z7=2 Read offset G54.
E1 contains X-axis offset.
E2 contains Z-axis offset.

Example 6 Calling a shift with angle of rotation of coordinate system

N100 G149 N1=54.02 X7=1 B47=2 Call shift G54.02
E1 has shift in X
E2 has angle of rotation of coordinate system

Example 7 Calling a zero point shift (G92/G93)

N100 G149 N1=92 X7=1 Z7=2 Call shift G92
E1 has shift in X
E2 has shift in Z
If a G92/G93 shift is called while G92/G93 is ineffective, the shift values 0 are obtained.

5.65 G150 Change tool data or zero offset values

To write values in the tool memory or zero offset memory within measuring cycle macros.

Format Tool data

To write data in tool memory

G150 T... {T2=...} {L1=...} {R1=...} {M1=...}

When T2=0, oversize (L4= or R4=) set to zero.

Better is to use G331.

To write tool status in tool memory:

G150 T... E...

Format Zero offsets

To write data in zero offset memory:

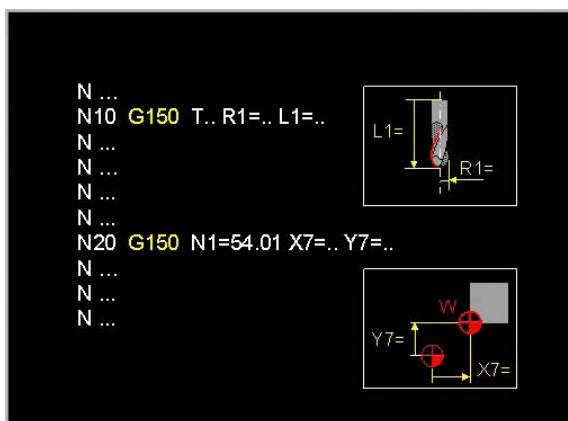
With standard zero offsets or MC84=0:

G150 N1=54...59 [(axis address)7=...] {(axis address)7=...} etc.

With MC84>0 zero offsets extends:

G150 N1=54.[nr] [(axis address)7=...] {(axis address)7=...} etc.

In G54.[nr] the number [nr] must be given in two digits (G54.01 and G54.10).



G Change tool or offset values
 T Tool number
 E E-parameter
 N1= Zero offset shift
 B47= Angle of rotation in B4=
 X7= Offset in X
 Y7= Offset in Y
 Z7= Offset in Z
 B7= Offset in B
 C7= Offset in C
 L1= Toollength value in T
 R1= Tool radius value in T
 T2= Tool offset index
 M1= Tool life value in T

Tool data

T	Tool number
T2=	tool offset index
E	E-parameter
L1=	Toollength value in T
R1=	Tool radius value in T
M1=	Toollife value in T

Zero offsets

N1=	Zero offset shift
X7=	Offset in X
Y7=	Offset in Y
Z7=	Offset in Z
A7=	Offset in A
B7=	Offset in B
C7=	Offset in C
B47=	Angle of rotation in B4=

Notes and usage

Associated Functions

G145, G148, G149

Tool number (T)

Number of tool for which tool data must be changed. If the FMS-tool memory (Flexible Manufacturing System) is in use, the complete number including the spare tool index has to be written.
The modal number of the actual tool is not influenced by this command.

Tool offset index (T2=)

A tool-offset index 0,1 or 2 can be specified. Default is T2=0. The offset index of the actual tool is not influenced by this command.

Tool status (E)

The tool status can be loaded from the indicated E-parameter into the tool memory.

Possible values for tool status are:

E... = 1	the tool is enabled and measured.
E... = 0	the tool is enabled but not measured
E... = -1	the tool is disabled
E... = -2	the tool time is expired
E... = -4	tool breakage error
E... = -8	the tool cutting force is exceeded
E... = -16	the tool time < T3 = programmed

A combination of error messages is also possible:
E... = -13 means: error message -8 and -4 and -2 and 1.

Zero offset (N1=)

Zero offset (G52, G54-G59 or **G54.[nr]**) that has to be changed.

Interrupt

The function cannot be stopped by an interrupt command.

Note: The function G150 is not permitted when G182 is active.
The tool data of T0 cannot be loaded.

Examples

Example 1 Write data in the tool memory.

N50 G150 T1 L1=E2 R1=4 M1=10 Change the data of tool No.1.
Store the value of parameter E2 as the tool length.
Make the tool radius equal to 4.
Make rest tool lifetime equal to 10 minutes.

When T2=0, the oversize (L4= or R4=) is set to zero.
Better is to use G331.

N50 G331 T1 I1 E2 writing length
N50 G331 T1 I4 E.. writing length oversize

Example 2 Write data in the zero offset memory.

N70 G150 N1=57 X7=E1 Z7=E6
or
N70 G150 N1=54.3 X7=E1 Z7=E6 Zero offset values of G57 are to be changed.
Store value of parameter E1 in G57 or G54.3 X-axis offset
Store value of parameter E6 in G57 or G54.3 Z-axis offset

Example 3 Changing a zero point shift with angle of rotation of coordinate system:

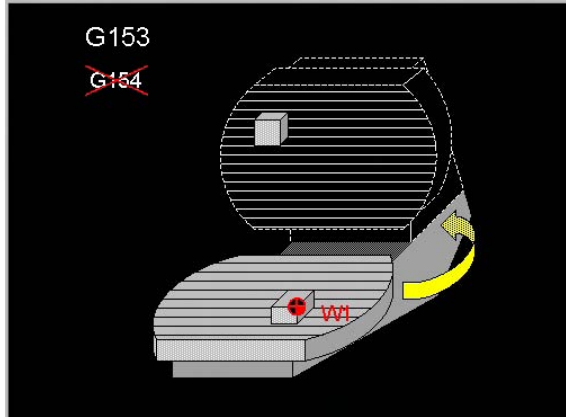
N70 G150 N1=54.03 X7=E1 B47=E6 Change zero point shift values of G54.03
Store value of E1 parameter in G54.3 shift in X
Store value of E6 parameter in G54.3 shift in B4=

5.66 G153 Correct workpiece zero point: OFF

G153 deactivates the zero point displacement. The active offset in the linear axes is cancelled.

Format

G153



G Correct workpiece zero point: OFF

Notes and usage

Modality

This function is mutual modal with G154.

Execution

G153 resets the modal status of the G154 function. The work piece zero point displacement is switched off.

G153 refrains from all actions until the movement in the previous block has ended (<INPOD>).

Display

The functions G153/G154 are displayed in the modal G row in the machining status display.

5.67 G154 Correct workpiece zero point: ON

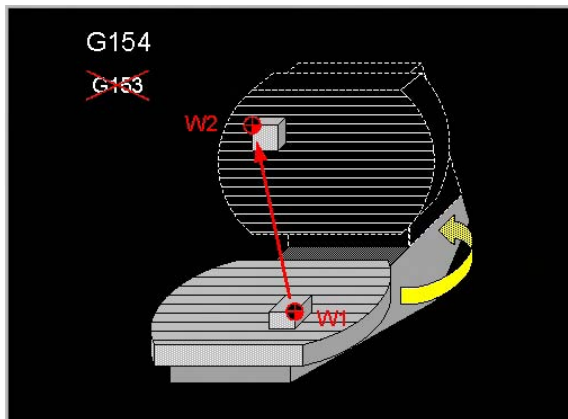
When the rotary axis rotates, the zero point from the work piece rotates with the work piece. The difference with G7 is, that the axes directions are not rotated also.

The G154-function activates the displacement of the work piece zero point by means of calculations in the kinematics. This can only be activated for rotary axes in the table. When active, the position of the programmed rotary axis is calculated in the position of the linear axes. The linear axes are not dragged along.

Note: The offset in the linear axes because of G108 is independent of G154/G153 and remains active. G108 has the same function, however is only active for the head.

Format

G154 {A1=..} {B1=..} {C1=..}



```
G    Correct workpiece zero point: ON
A1=  ZPS A-axis (0=not, 1=settle)
B1=  ZPS B-axis (0=not, 1=settle)
C1=  ZPS C-axis (0=not, 1=settle)
```

A1= Defines whether the position of the A-axis in the table is calculated in the linear axes.
 0 = not calculated (default)
 1 = calculated
 This address is only allowed when there is an a-axis in the table.
 B1= and C1= for the B-axis and C-axis.

Default settings

When no address is programmed all axes in the table are activated.

Notes and usage

Modality

This function is mutual modal with G153.

Execution

When G154 is active, the display of the linear axes at the end of every positioning of the axes defined in G154 is adapted.

G154 refrains from all actions until the movement in the previous block has ended (<INPOD>).

Switching off G154

The function G154 is switched off by G153.

After <cancel program>, M30, <Clear Control> or switching on the CNC, the function G154 remains active. The programmed rotary axis is saved in the stand-by memory.

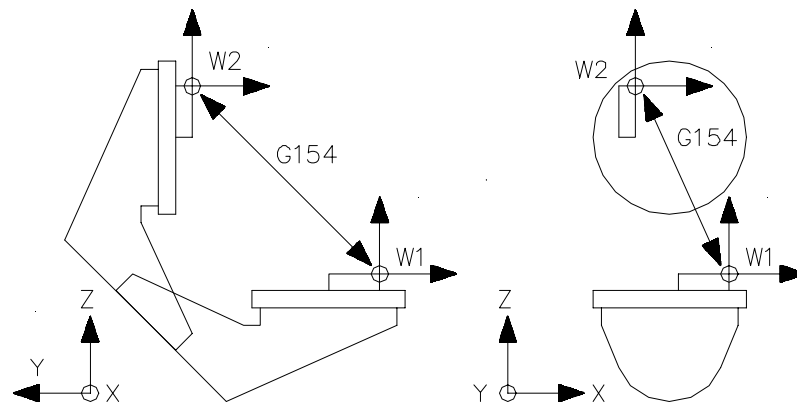
Interrupt

When a rotary axis movement is interrupted, the display of the linear axes is not adapted.

Only after <Emergency Stop>, <cancel program> or <manual mode> during program interrupt, the display of the linear axes is updated to the state of the rotary axis.

Manual mode

The function G154 remains active after M30 and is active in manual mode. The display of the linear axes is updated when the rotary axis movement is stopped.



W1 = Work piece zero point in position 1
W2 = Work piece zero point in position 2.
In this case the table is rotated 180° around the B-axis.
G154 is the zero point displacement caused by the axis rotation.

Zero point shift

A zero point shift (G54, G92, G93) or IPLC-shift in the relevant rotary axis is taken into account. This means that the new zero point of the rotary axis is taken as the zero position for the kinematic calculations.

Status-display

The G153- / G154-status is displayed in the modal G-group display.

Example Activating zero point displacement.

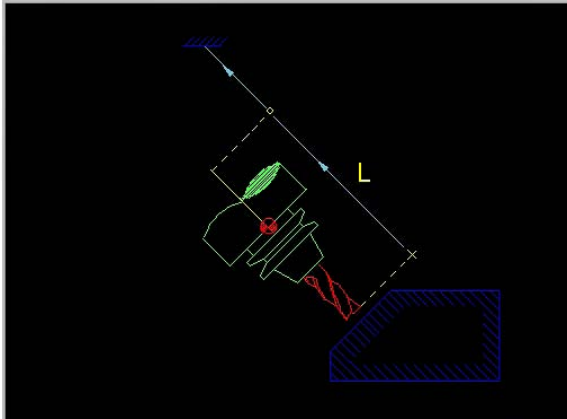
Programming example	Description
N10 G154 B1=1	Work piece zero point is corrected after the table rotation.

5.68 G174 Tool withdrawal movement

Movement to move the tool axis clear during 5-axis milling.

Format

G174 {L....} {X1=.. or Y1=.. or Z1=..}



```
G    Tool retract movement
L    Retract distance
X1=  1=Retraction only in this axis
Y1=  1=Retraction only in this axis
Z1=  1=Retraction only in this axis
```

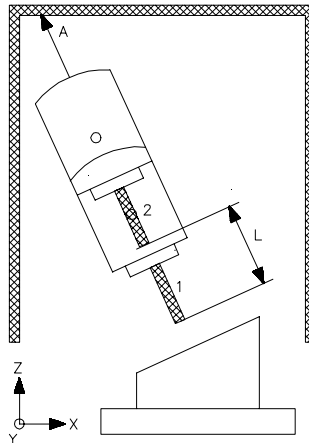
Notes and usage

Execution

With this function, you are always able to move away in the direction of the tool axis. The tool is withdrawn until the 'first' SW limit switch is reached. The direction of movement is determined by the position of the milling head. In the direction of the tool head the tool is withdrawn.

Execution (X1= or Y1= or Z1=)

With programming an X1= or Y1= or Z1= will be fixed, which machine axis will be moved. During G7 the machine axis can be different from the programmed axis. A combination of X1=, Y1= und Z1= is not allowed (P414). **The movement is not in the tool direction.** X1=1 means, that the X-axis will be moved.



- 1 Starting position
- L Withdrawal distance
- 2 End position
- A Limitation by software limit switch

Withdrawal distance (L)

The withdrawal distance ($L > 0$) defines the distance travelled in the direction of the tool.

An error message is given, when L is bigger than the distance to the software limit switch (Z31).

Without programming the withdrawal distance (L) the movement is limited by the software limit switch.

Execution (G0)

G174 is executed in rapid. If F6= is programmed this feed is taken.

Following G107, G0 or G1 from the previous block is modally active again.

Example Tool withdrawal movement.

N10 G174 L100

Tool retracts 100 mm.

N..

N30 G174 L100 X1=1

Tool moves 100 mm in the X-axis.

5.69 G180 Basic coordinate system

This function has two meanings:

- 1) Deactivating cylindrical coordinates systems (G182).
- 2) Defining of mainplane and tool axis (Basic coordinates system).

Format

G180: Basic coordinate system

G180 [auxiliary axis 1][auxiliary axis 2][Tool axis]

```
G    Cancel cylinder interpolation
X    1=allocate axis to coord.system
Y    1=allocate axis to coord.system
Z    1=allocate axis to coord.system
B    1=allocate axis to coord.system
C    1=allocate axis to coord.system
```

General principles

The normal expression is G180 X1 Y1 Z1

The only following configurations are possible:

Auxiliary axis 1	X
Auxiliary axis 2	Y
Tool axis	Z or W

The correct procedure depends on 3 different types of information:

- 1) The tool axis is determined by G17/G18/G19 (G17 Z).
- 2) G180 determines which axes are to be substituted. (G17 W in Z)
- 3) The machine constants for the tool axis definition should also be correct (tool axis W belongs to Z).

Notes and usage

Modality

G180 and G182 are modal functions

Functions to be cancelled

The functions G41-G44, G64, G73, axis rotation (G92/G93 B4=) and G141 must be cancelled before G180 is activated.

Any other function, which is active immediately before the G180 block, remains active.

Note: The words X, Y, Z cannot be programmed without any value. Therefore the value 1 is written to it. This value has no meaning.

Radius and tool length compensation

The tool length compensation is active in the defined tool axis. The radius compensation is active in the auxiliary axis.

Machine constant

The machine constant must be accurate. If W-Axis is the fourth axis then MC117 = 3 (just like Z-Axis). MC3401 = 0 (W-Axis is a Linear axis).

Coordinates

Only Cartesian coordinates can be used.

Note: If G180 is programmed and radius compensation is still active, the compensation is cancelled with the G180. It is advised to cancel radius compensation with G40 and then to return to the basic coordinate system.

Cancellation

Cylinder interpolation is cancelled by either the G180 function or CLEAR CONTROL.

Default mode

When turning on the controller or activating Softkey CLEAR CONTROL G180 X1 Y1 Z1 is automatically turned on.

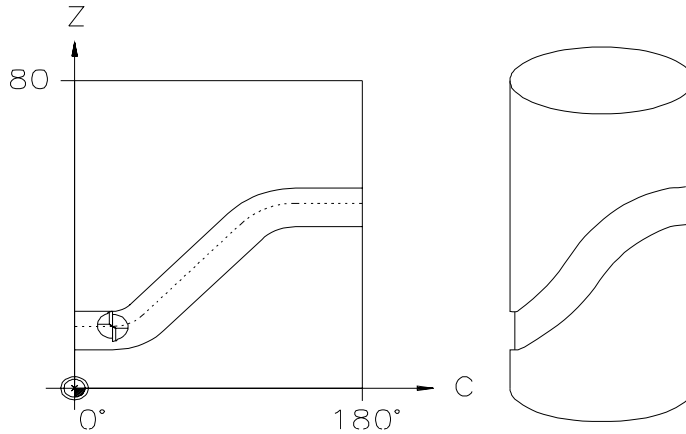
Example

N12340	
N1 G17 S1000 T1 M6	Define plane of operation.
N2 G54	Activate a stored zero offset.
N3 G180 X1 Y1 W1	Activate auxiliary plane XY and Tool axis W.
N4 G81 Y2 B10 Z-22 F1000 M3	Define drilling cycle.
N5 G79 X0 Y0 Z0	Drilling hole with the feedrate active in the W axis.

5.70 G182 Cylindrical coordinate system

Selection of the cylindrical coordinate system. Using this system, you can easily program contours and positions on the curved surface of a cylinder.

Format



G182 Cylindrical coordinate system

To activate the cylindrical coordinate system

G182 [cylinder axis] [rotary axis] [rotary axis] R..

G182 R..

Format with active G182

Rapid

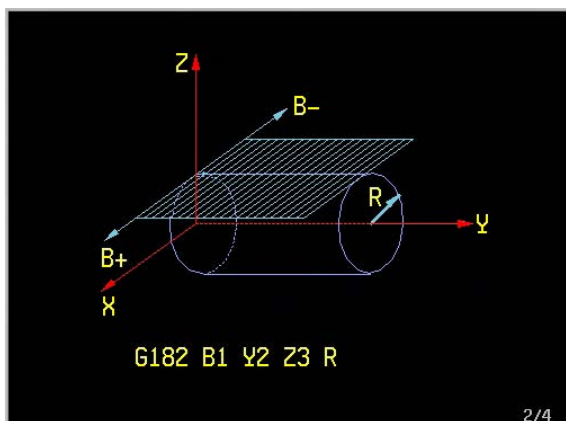
G0 [cylinder axis] [rotary axis] {rotary axis}

Linear movement:

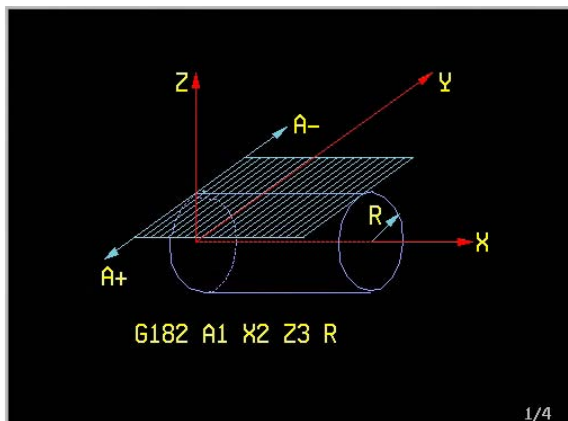
G1 [cylinder axis] [rotary axis] {rotary axis} {F..}

Circular movement:

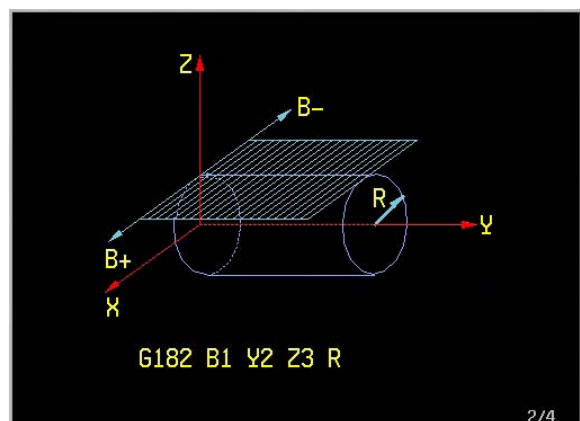
G2/G3 [cylinder axis] [rotary axis] R..



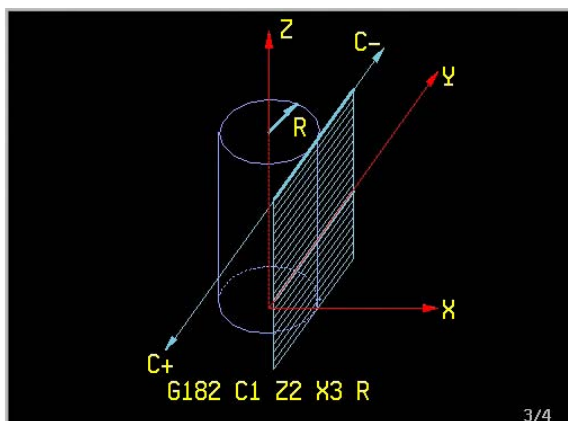
```
G  Activate cylinder interpolation
X  Cylinder plane:2 / Tool axis:3
Y  Cylinder plane:2 / Tool axis:3
Z  Cylinder plane:2 / Tool axis:3
B  Cylinder plane:1
C  Cylinder plane:1
R  Cylinder radius
```



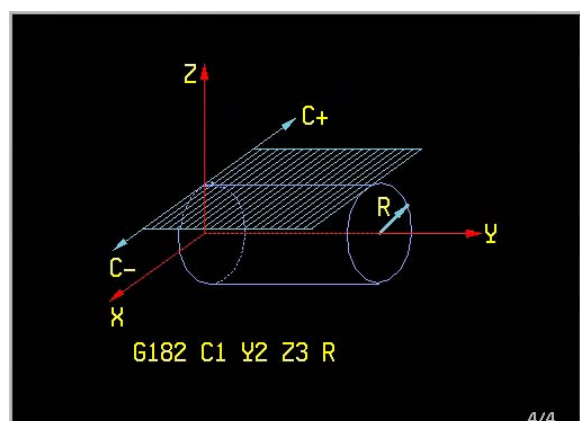
G182 A1 X2 Z3 R..
or (as previous)
G182 A1 X1 Z1 R..



G182 B1 Y2 Z3 R..
or (as previous)
G182 B1 Y1 Z1 R..



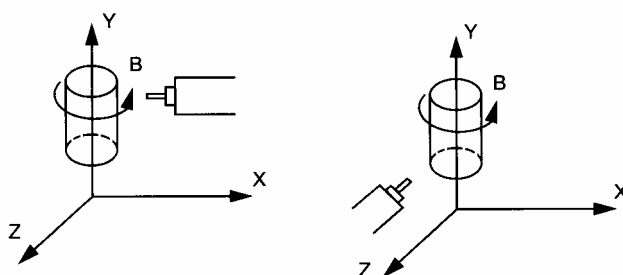
G182 C1 Z2 X3 R..
or (as previous)
G182 C1 X1 Z1 R..



G182 C1 Y2 Z3 R..

General principles

Contours on the curved surface of a cylinder are drawn in a plane representing the curved surface. This plane is defined with the rotary axis, the cylinder axis and the cylinder radius. In this plane linear and circular movements with radius compensation can be programmed. During the execution of the program these movements are converted to movements with a linear (= the axis of the cylinder) and a rotary axis (= the axis rotated about the cylinder axis). This is called cylinder interpolation.



In general the following configurations are possible:

Rotary axes	A,	B,	C
Cylinder axes	X,	Y,	Z
Tool axis	Y or Z,	X or Z,	X or Y.

The BY-plane for cylinder interpolation

The correct procedure depends on 3 different types of information:

- 1) The tool axis is determined by G17/G18/G19 (G17 Z).
- 2) G182 determines which axes are to be substituted. (G17 AX or BY)
- 3) The machine constants for the rotary axis definition should also be correct. (Rotary axis A belongs to X).

Notes and usage

Modality

G180 and G182 are modal functions

Functions to be cancelled

The functions G41-G44, G64, G73, axis rotation (G92/G93 B4=) and G141 must be cancelled before G182 is activated.

Any other function, which is active immediately before the G182 block, remains active.

Specifying the cylinder plane

The words X, Y, Z, A, B, C cannot be programmed without any value.

In the G182-block the configuration for cylinder interpolation is programmed:

Standard configuration

Rotary axis:	A1	B1	C1
Cylinder axis:	X1	Y1	Z1
Tool axis:	Y1/Z1	X1/Z1	X1/Y1
Cylinder radius:	R	R	R

Other configuration

Rotary axis marked with 1:	A1	B1	C1
Cylinder axis marked with 2:	X2Y2Z2	Y2X2Z2	Z2X2Y2
Tool axis marked with 3:	Y3Z3X3	X3Z3Y3	X3Y3Z3
Cylinder radius:	R	R	R

Note: When the mark is 1 then only a standard configuration is possible.

Machine constants

The machine constants must be accurate.

MC 102 = 1, MC103 = 88 (X-axis)

MC 107 = 2, MC108 = 89 (Y-axis)

MC 112 = 3, MC113 = 90 (Z-axis)

MC 117 = 4 belongs to axis 1 (4-3), MC118 = 65 (A-axis turns around X-Axis)

MC 122 = 6 belongs to axis 3 (6-3), MC123 = 67 (C-axis turns around Z-Axis)

Default cylinder plane

When a machine tool has only one rotary table, the configuration for cylinder interpolation is defined in the Machine Constants. Therefore, if the axis configuration is not programmed in a G182-block, these settings are used automatically by the CNC.

Cylinder radius

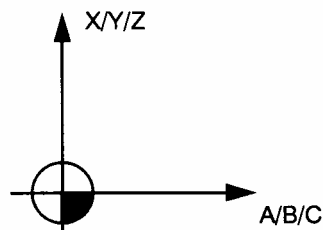
The cylinder radius is used by the CNC for calculating the feedrate of the rotary axis. The radius value must be between 1 mm and 500 mm. If the R-word is not programmed in a G182-block, an error message is displayed.

Changing the cylinder radius

The radius of the cylinder can be changed with the R-word in another G182-block. In this block the definition of the plane has to be repeated.

Plane of the curved surface

The plane of the curved surface is called either AX, BY or CZ, depending on which cylinder the contour has to be made.



Axes of the plane of the curved surface

The horizontal axis is the rotary axis and is programmed with the corresponding axis address A, B, C in degrees and decimal parts thereof.

The vertical axis is the cylinder axis and is programmed with the corresponding axis address X, Y, Z in mm or inches.

The tool axis is perpendicular to the wall of the cylinder, programmed in mm or inches with the addresses Y or Z for the AX-plane, X or Z for the BY-plane, X or Y for the CZ-plane depending on which axis the tool is loaded.

Datum point

The datum point in the rotary, cylinder and tool axis must be programmed, before cylinder interpolation is activated. This can be achieved with
 G51-G52 (p) reset axes.
 G53-G59 or G54I[nr.] stored zero offset
 G92/G93 a datum point shift

Once cylinder interpolation (G182) is activated a datum point shift is not allowed until the basic coordinate system (G180) is chosen again.

Coordinates

Only Cartesian coordinates can be used.

The functions G90 and G91 are used for programming absolute (G90) or incremental (G91) dimensions and can be used with the rotary, cylinder and tool axis.

Rapid movements (G0)

A rapid traverse movement is programmed with G0 and the end point of the movement. Two or three axes can be programmed in one block. The axes move with the positioning logic:

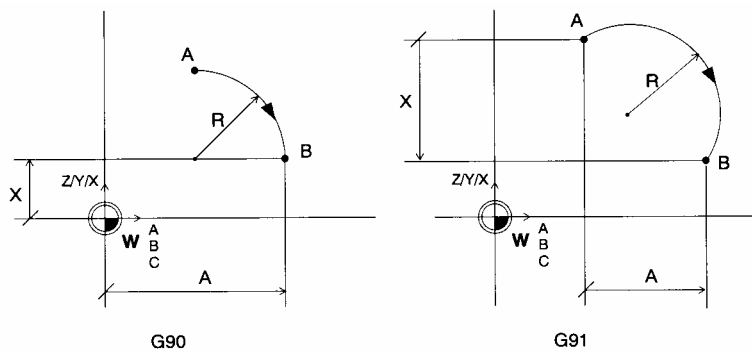
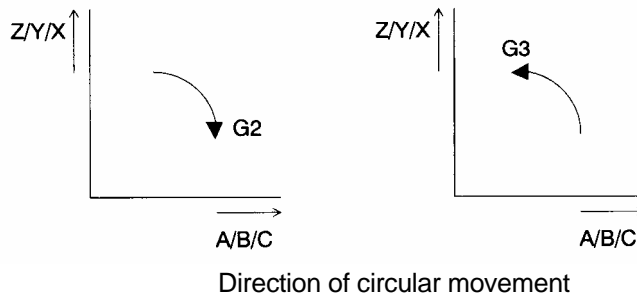
- | | | |
|----------------------------|----|-----------------------|
| Tool towards the cylinder: | 1. | movement in the plane |
| | 2. | tool axis movement |
| Tool from the cylinder: | 1. | tool axis movement |
| | 2. | movement in the plane |

Linear feed movements (G1)

A linear feed movement is programmed with G1, the end point of the movement and the feedrate. Two or three axes can be programmed in one block. All axes move simultaneously and reach their end point at the same time.
 The programmed feedrate is the surface feed on the cylinder at the radius from the G182-block.

Circular feed movements (G2/G3)

A circular feed movement can only be programmed with a G2 or G3, the end point coordinates and the radius of the arc (R-word).



Radius compensation

For radius compensation in the plane of the cylinder the functions G40, G41, G42, G43 and G44 can be used. These functions have the same meaning as in the basic coordinate system (G180 active). For defining LEFT and RIGHT one should look from the tool to the cylinder.

Note: If G180 is programmed and radius compensation is still active, the compensation is cancelled with the G180

It is advised to cancel radius compensation with G40 and then to return to the basic coordinate system.

Tool size

From the control side of view there is no restriction on the size of the tool radius. However, if the tool radius is too large, undercuts may be produced. These undercuts depend on the shape and size of the tool and the depth of operation.

Note: With contours on the cylinder the greatest accuracy is achieved with a tool of which the diameter is about 0.2 mm less than the width of the groove.

Cancellation

Cylinder interpolation is cancelled by either the G180 function or CLEAR CONTROL.

Default mode

G180 is made active automatically when the CNC is switched on, or the CLEAR CONTROL operation is performed.

Functions permitted

G0, G1, G2/G3, G4, G14, G22, G23, G29, G40-G44, G90/G91, G94/G95, G180/G182

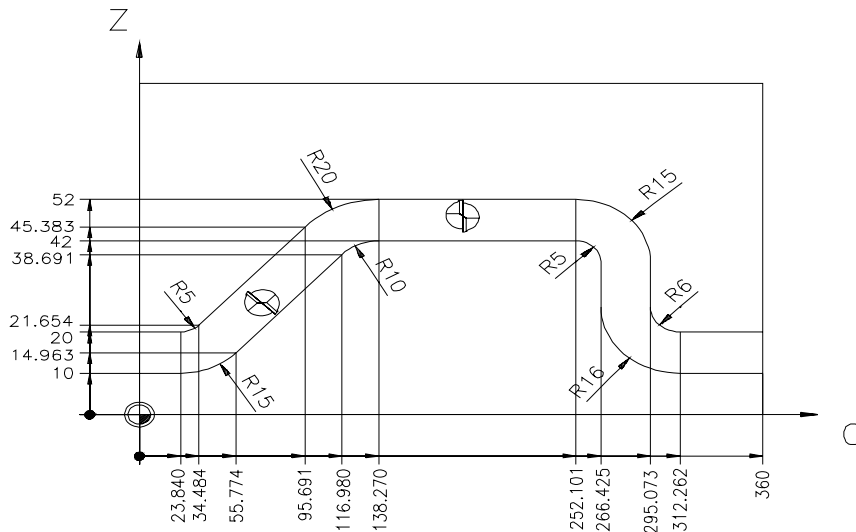
- Notes
1. If G14 or G29 are used during cylinder interpolation, the target block number for the jump must be in the program section for cylinder interpolation.
 2. The function G94 or G95, which is active at activating cylinder interpolation, is not influenced by G182.
 3. All other G-functions are not permitted, when G182 is active.

Examples

Example 1

The groove on the curved surface of a cylinder of diameter=40 mm has to be milled with a slotting end mill D=9.5 mm. The depth of operation=4 mm. The workpiece is machined horizontal with the rotary axis C, the cylinder axis Z and tool axis Y.

The contour is programmed in this example.



N12340

N1 G18 S1000 T1 M6

Load the tool. Select plane. Because the Y-axis is the tool axis, G18 must be chosen

N2 G54

Activate the stored zero offset of G54

N3 G182 C1 Y2 Z3 R20

Use the cylindrical coordinate system for the BY-plane, the Y-axis as tool axis and a cylinder radius of 20 mm.

N4 G0 Z15 C0 Y22 M3

Move to the start position and activate the spindle

N5 G1 Y16 F200

Feed movement to depth

N6 G43 Z10

Radius compensation T0 the lower contour

N7 G41

Activate radius compensation LEFT

N8 G1 C23.84

Points of the lower contour till 365° (=360° +run out)

N9 G3 Z14.963 C55.774 R15

N10 G1 Z38.691 C116.98

N11 G2 Z42 C138.27 R10

N12 G1 C252.101

N13 G2 Z37 C266.425 R5

N14 G1 Z26

N15 G3 Z10 C312.262 R16

N16 G1 C365

N17 G40

Cancel radius compensation; the tool tip should come at the position 365°.

N18 G41 Z20

Activate radius compensation LEFT and move to upper contour

N19 G1 C312.262

Points of the upper contour

N20 G2 Z26 C295.073 R6

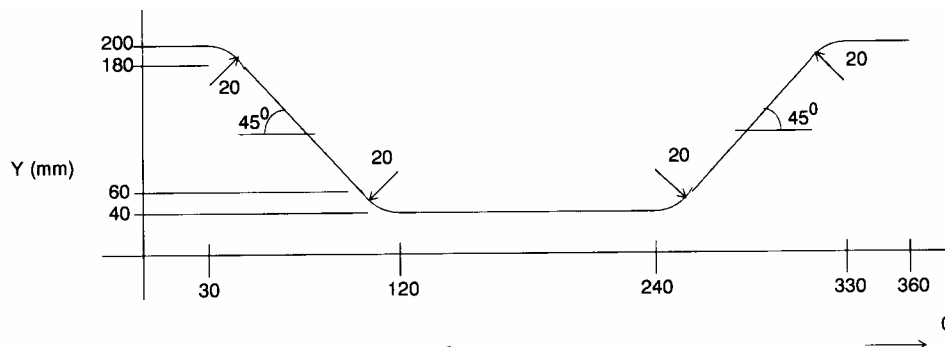
N21 G1 Z37

N22 G3 Z52 C252.101 R15
 N23 G1 C138.27
 N24 G3 Z45.383 C95.691 R20
 N25 G1 Z21.654 C34.484
 N26 G2 Z20 C23.84 R5
 N27 G1 C0
 N28 G40
 N29 G180
 N30 G0 Y100 M30

Cancel radius compensation
 Return to the basic coordinate system of the machine tool
 Retract tool from part and end of program.

Example 2

The upper part of a groove on the curved surface of a cylinder with a radius of 114.6 mm is drawn. The workpiece is machined horizontal with the rotary axis B, the cylinder axis Y and tool axis Z. The contour is programmed in this example.



N9011
 N1 G17
 N2 G54
 N3 S500 T1 M6
 N4 G182 B1 Y2 Z3 R114.6

N5 G0 Y180 B0 Z116 M3
 N6 G43 Y200
 N7 G1 Z114 F300
 N8 G42
 N9 B30

N10 G2 Y194.142 B37.071 R20
 N11 G1 Y45.858 B112.929
 N12 G3 Y40 B120 R20
 N13 G1 B240
 N14 G3 Y45.858 B247.071 R20
 N15 G1 Y194.142 B322.929
 N16 G2 Y200 B330 R20
 N17 G1 B360
 N18 G40
 N19 G0 Z150 M30

Set the plane for operation
 Activate the stored zero offset from G54
 Load tool 1 and activate the spindle
 Use the cylindrical coordinate system for the BY-plane, the Z-axis as tool axis and a cylinder radius of 114.6 mm.
 Rapid tool movement to start position and start the spindle
 Move tool to the contour
 Feed movement at depth
 Set radius compensation RIGHT
 Move along the contour. The contour points must be calculated by the part programmer from the data given on the drawing.

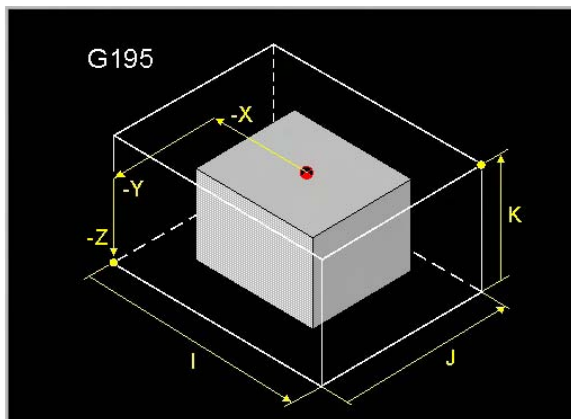
Cancel radius compensation
 Retract tool from part and end of program.

5.71 G195 Graphic window definition

To define the dimensions of a 3D graphic window and its position relative to the zero point W. In this window the workpiece and machine parts can be situated for a graphical simulation of a part program run.

Format

G195 X... Y... Z... I... J... K... {B...} {B1=...} {B2=...}



```
G  Graphic window definition
X  Start point coordinate
Y  Start point coordinate
Z  Start point coordinate
B  Rotation around hor. axis (3D)
I  Dimension parallel to X
J  Dimension parallel to Y
K  Dimension parallel to Z
B1= Rotation around vert. axis (3D)
B2= Rotation around third axis (3D)
```

Notes and usage

Associated functions

G98, G99, G196 to G199

Graphical support

Refer to the appendix GRAPHICAL SUPPORT at the end of this manual for a short overview about the graphical support provided in the CNC PILOT control system and to the user manual for using the graphical support.

Graphic window

The window, thus a bounded area on the display, is a rectangular 3D box which dimensions are defined by the G195-function.

The window is used with the graphical simulation, but also with the synchron graphics with which the actual tool movements on the machine can simultaneously be seen on the display of the control.

Relevant window axis

Because the display on the control is a rectangle, the scale on the shorter side (Y-axis in XY-plane), which is calculated from the programmed value (the J-word), also determines the scale on the longer axis (X-axis in XY-plane).

Contour definition (G196 - G199)

Besides a window also an outer contour of a workpiece blank and/or machine parts and, if required, an inner contour can be defined for the graphical simulation. The dimensions of these contours are programmed with the functions G196 to G199. Refer to these functions for defining a contour.

Default window dimensions

If the dimensions of the 3D window are not defined the CNC uses the software limit switches' distances as default values.

Angle of viewing (B, B1=, B2=)

With the synchron graphics or the 3D-wire plot the workpiece can be seen rotated. The angles for viewing the rotated workpiece on the display are defined by the words B, B1= or B2=.

	B rotation about	B1= rotation about	B2= rotation about
XY- plane (G17)	X- axis	Y- axis	Z- axis
XZ- plane (G18)	Z- axis	X- axis	Y- axis
YZ- plane (G19)	Y- axis	Z- axis	X- axis

Other methods are available for selecting an angle of viewing and are described in the user manual.

Default settings for angles of viewing

If the angles of viewing are not programmed the following default settings are automatically used by the control: B60, B1=30 and B2=0

Restrictions

The function G195 is not permitted in MDI or the TEACH-IN (PLAYBACK) mode.

Example

N9000	
N1 G17	Define the machining plane
N2 G195 X-30 Y-30 Z-70 I170 J150 K100	Define the graphic window
N3 G199	Start of the contour description section

5.72 G196 End contour description

To end the contour description for the graphical simulation of a partprogram run.

Format

G196

Notes and usage

Associated functions

G98, G99, G195, G197 to G199

Functions active after a G196 block

After a G196 the function G64 is reset, so G63 is activated again.

If the last function of the contour description is a G2 or G3, this function is set back to the active function G0 or G1 before the G199.

All other modal functions, which are active before the G199, are not influenced.

Restrictions

1. A G199 must be programmed before the G196. If not, an error message is displayed.
2. The G196 function cannot be used in MDI and the TEACH-IN (PLAYBACK) mode.

Example

N9000	
N1 G17	Define the plane of operation
N2 G195 X-30 Y-30 Z-70 I170 J150 K100	Define graphic window
N3 G199 X0 Y0 Z0 D-20	Start graphic contour description
...	
N10 G196	End graphic contour description.

5.73 G197/G198 Begin inner/outer contour description

To define the start point in a graphical presentation of a contour from a blank workpiece or a machine part. Outer as well as inner contours can be defined. After defining the start point of the contour the contour itself can be programmed with the functions G1 and G2 or G3.

G197: define the start point of an inner contour

G198: define the start point of an outer contour

Format

To define the start point of an inner contour

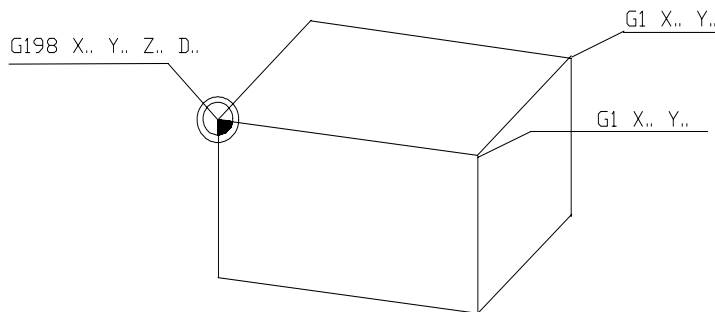
G197 X... Y... D...

To define the start point of an outer contour

G198 X... Y... {Z...} D...

To define the start point of a geometry element:

G198 X... Y... {Z...} {I1=..}.



Start point for defining the blank contour.
zero point defined by G199.

```
G   Begin inside contour description
X   Startpoint coordinate
Y   Startpoint coordinate
D   Depth inside contour
?90= Startpoint abs. (X,Y,Z..)
?91= Startpoint incr. (X,Y,Z..)
I1= Colour
```

G197

```
G   Begin outside contour description
X   Startpoint coordinate
Y   Startpoint coordinate
Z   Startpoint coordinate
D   Depth outside contour
?90= Startpoint abs. (X,Y,Z..)
?91= Startpoint incr. (X,Y,Z..)
I1= Colour
```

G198

Notes and usage

Associated functions

G98, G99, G195 to G196, G199

Start point of contour description

The start point of the contour is programmed with the linear axes coordinates X, Y and Z. These coordinates are related to the point, programmed in a G199 block.

The coordinates of the start point must be absolute, Cartesian coordinates. Polar coordinates are not allowed.

If the Z-word is not programmed in a G198 block, Z0 is used as a default setting.

Note The Z-word is not used with the G197.

Contour description

Once the start point of the contour is established (in a G197 or G198),
The contour is programmed with the functions G1, G2 and G3.

G1 a line with its end point.

G2/G3 a circular arc with end point and radius or centre point and end point.
a complete circle with centre point only.
using the helix or 2.5 D interpolation is not allowed.

An end point or centre point can be programmed with absolute Cartesian or polar coordinates. They are related to the point programmed in the G199 block.

For complicated contours the geometry of the control (G64) can be used.

The contour must be closed, otherwise, a straight line will be generated automatically from the end point to the start point.

The contour must lie in the main plane defined by the active function G17, G18 or G19.

The depth of the contour (D)

The depth of the outer contour is programmed with the D-word. Its value is related to the tool axis coordinate of the G198 block.

If the bottom of the contour is Z0 (XY-plane), the depth is positive. If the upper surface is Z0, the depth is negative.

The depth of the inner contour is also programmed with the D-word. Its value is related to the depth of the outer contour.

Inner contours

An inner contour must lie within a previously defined outer contour.

Inner contours may not intersect the sides or be tangent to the sides of the outer contour.

An inner contour cannot be inside another inner contour.

More than one contour description

If a part is composed of independent contours, e.g. layers or holes, it is possible to define each contour separately with the functions G198 and G197. With one of these functions the previous contour description is ended and the description of the next contour starts.

A complete contour, thus the outer as well as the inner contour, must be programmed in one section.

If all contours are defined the function G196 is used to end the complete contour description.

Drawing geometric elements (line or circle)

When the drawn function is activated (G199 B4) in a G198 block a colour can be defined. The next geometric elements can be programmed in 3D.

The Depth (D) has no meaning and is not permitted.

Only G0, G1, G2 and G3 are permitted.

More geometric elements can be programmed in succession. In a partprogram more groups of geometric elements can be defined.

The contour is only visible during simulation in wireplot graphic.

Possible colours (I1=):

1	red	11	light red
2	green	12	light green
3	yellow	13	light yellow
4	blue	14	light blue
5	grey	15	light magenta
6	cyan	16	light cyan
7	white	17	bright white
8	black	18	black
9	foreground	19	foreground
10	background	20	background

To delete a geometric element, the same geometry in black-ground colour must be drawn.

Ending a contour description

A contour description is finished with either a function G198 for defining another outer contour, a G197 for defining an inner contour or a G196 indicating that no more contours are followed.

Contour description in macro

If a macro is to be used for describing a contour, e.g. for describing machine parts, all graphic functions (G199, G198, G197 and G196) must be programmed in the macro.

Functions allowed with a contour description

With a contour description only the following functions are permitted:

- G1, G2, G3: for describing a contour
- G64/G63: for defining the contour with the geometry function
- G196: for ending the contour description
- G197/G198: for defining another contour

Functions ignored with the contour description

If radius compensation, scaling, mirror image or axis rotation is activated before the G199, the function is ignored during the graphical simulation of the contour. No error message is displayed to this effect.

Therefore it is advised to activate the following functions before the G199:

- G40 (no radius compensation)
- G72 (no mirror image or scaling)
- G90 absolute coordinates
- G93 B4=0 (no axis rotation)
- G180 Basic coordinate system

Restrictions with the contour description

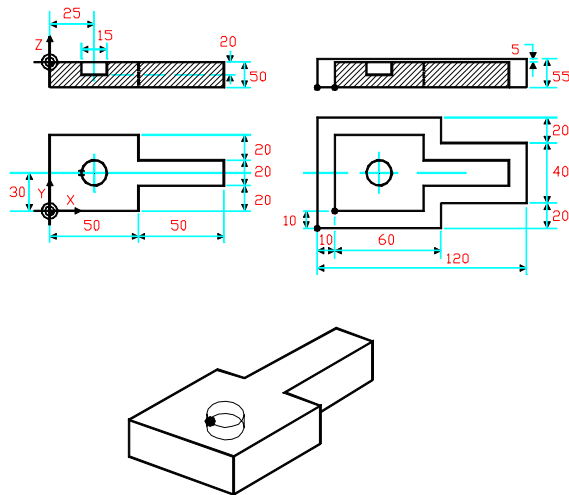
1. The axes coordinates must be absolute and lie in the active main plane.
2. Polar coordinates and the combination of one coordinate and an angle can be used with the G1 and G2/G3 blocks. They cannot be used to define the start point of the contour.
3. Previously defined points cannot be used with a contour description.

Restrictions with the use of G197 and G198

1. With G197 the Z-coordinate is not used. The depth value (D-word) of the G197 is related to the depth of the outer contour.
2. The functions G197 and G198 are not permitted in MDI or in the TEACH-IN (PLAYBACK) mode.

Example

Example 1 Outer and inner contour definition



N1971981

N1 G17

N2 G195 X-10 Y-10 Z10 I120 J80 K-40

N3 G199 X0 Y0 Z0 B1 C2

N4 G198 X0 Y0 Z0 D-20

N5 G1 X50

Set the plane of operation to be the XY-plane

Set the graphic window to define the 3D space

Start of contour description section.

Define start point of outer contour.

Description of the outer contour. The coordinates are related to the point programmed in the G199 block.

N6 Y20

N7 X100

N8 Y40

N9 X50

N10 Y60

N11 X0

N12 Y0

N13 G197 X17.5 Y30 D-10

N14 G2 I25 J30

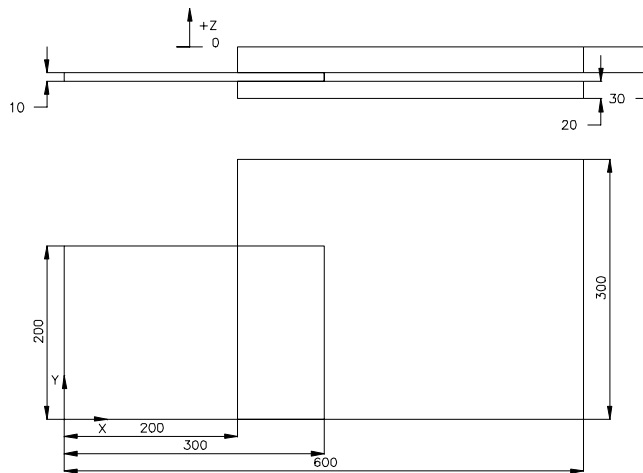
N15 G196

Define start point of inner contour.

Describe the inner contour to be a complete circle.

End of contour description section

Example 2 Contour with different layers



N1971982

N1 G54

N2 G17

N3 G195 X-5 Y-5 Z5 I610 J310 K-70

N4 G199 X0 Y0 Z0 B1 C2

N5 G198 X0 Y0 Z-30 D-10

N6 G1 X300

N7 Y200

N8 X0

N9 Y0

N10 G198 X200 Y0 Z0 D-30

N11 G1 X600

N12 Y300

N13 X200

N14 Y0

N15 G198 X200 Y0 Z-40 D-20

N16 G1 X600

N17 Y300

N18 X200

N19 Y0

N20 G196

Set the program zero point

Set the plane of operation to be the XY-plane

Set the graphic window to define the 3D space

Start of contour description section.

Define start point of outer contour (first layer).

Describe the first layer

Define start point of outer contour (second layer).

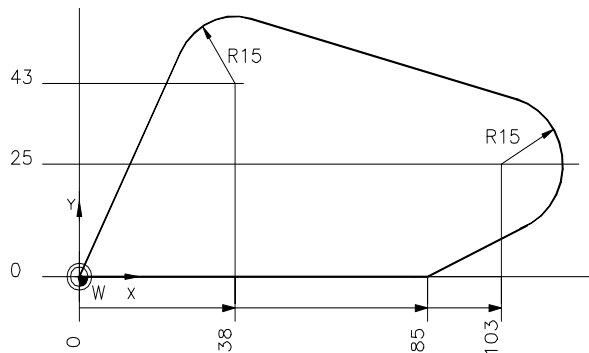
Describe the second layer

Define start point of outer contour (third layer).

Describe the third layer

End of contour description section

Example 3 Contour described with geometry



N1971983

N1 G54

N2 G17

N3 G195 X-10 Y-10 Z10 I138 J78 K-30

N4 G199 X0 Y0 Z0 B1 C2

N5 G198 X0 Y0 Z0 D-10

N6 G64

Set the program zero point

Set the plane of operation to be the XY-plane

Set the graphic window to define the 3D space

Start of contour description section.

Define start point of outer contour.

Describe the outer contour. The coordinates are related to the point programmed in the G199-block. The geometry of the control is used to program the contour.

N7 G1 X85 Y0

N8 R1=0

N9 G3 I103 J25 R15 R1=0

N10 G1 R1=0

N11 G3 I38 J43 R15 R1=0

N12 G1 X0 Y0

N13 G63

N14 G196

N15 S1000 T1 M6

N16 G0 X-15 Y-15 Z20 M3

N17 G1 Z-10 F1000

N18 G43 Y0

N19 G42

N20 G14 N1=6 N2=13

End of contour description section

Load the tool and set the spindle speed

Start the spindle and move the tool to the start point.

Move tool to depth

Move tool to the contour

Set radius compensation RIGHT

Move along the contour. With the G14 function the programmed contour in the outer contour description is used.

Cancel radius compensation

Move tool to the start point and away in the tool axis. End of program

N21 G40

N22 G0 X-15 Y-15 Z100 M30

Example 4 Draw a raw material (cylinder) for turning.

N9999

N1 G17

N4 G36 (endless turning)

N6 G17 Z1=1 Y1=2

Set the plane for milling. Length compensation in Z-direction

Turning mode

Set the plane for turning.

Mainaxis-1 is Z, Mainaxis-2 is Y.

Radius compensation in ZY plane.

Set the graphic window

Start of contour description section. B4 means draw.

Define start point of contour. I1=14 means colour pale blue

Upper circle of the cylinder.

Connection line

Lowest circle of the cylinder.

End of contour description section

N7 G195 X-1 Y-1 Z1 I2 J12 K-11

N8 G199 X0 Y0 Z0 B4 C2

N9 G198 I1=14 X0 Y8 Z0

N10 G2 X0 Y0 I0 J0

N11 G1 X0 Y8 Z-8

N12 G2 X0 Y8 I0 J0

N13 G196

5.74 G199 Begin contour description

1. To define the position of the workpiece blank contour related to the program zero point or machine zero point. This position is used during the graphic simulation of the program run.
2. To define the position of any machine part with which the tool might collide. Collision to be detected during the graphic simulation.
3. Drawing a contour during the wireplot simulation.
4. To draw one or more geometry elements (line or circle) during the wireplot simulation.

Format

To define a blank contour

G199 [Coordinates of position] B1 {C1} {C2}

To define a machine part

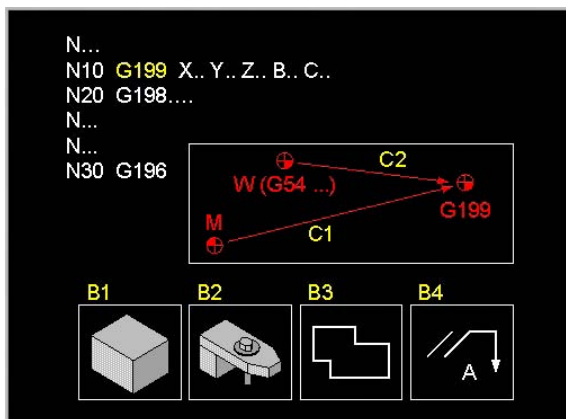
G199 [Coordinates of position] B2 {C1} {C2}

To draw a contour during the wireplot simulation.

G199 [Coordinates of position] B3 {C1} {C2}

Draw one or more geometry elements (line or circle) during the wire model graphic simulation.

G199 [co-ordinates of position] B4 {C1} {C2}



```
G  Begin graphic model description
X  Startpoint coordinate
Y  Startpoint coordinate
Z  Startpoint coordinate
B  1=mat., 2=mach., 3=contour, 4=draw
C  Zero point 1=Machine, 2=Workpiece
?90= Startpoint abs. (X,Y,Z..)
?91= Startpoint incr. (X,Y,Z..)
```

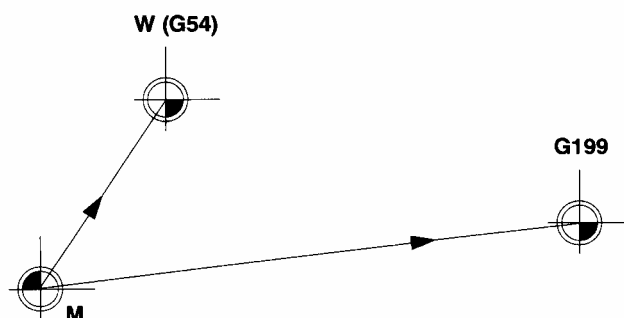
Notes and usage

Associated functions

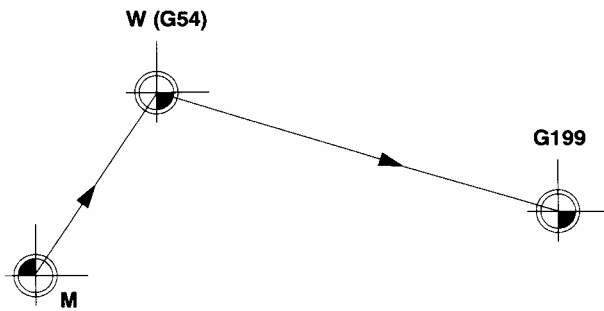
G98, G99, G195 to G198

Zero point (C1/C2)

The position of the contour can be related to either the machine zero point M_0 or the program zero point W.



The contour position is related to the machine zero point M_0 . (C1)



the contour position is related to the program zero point W. (C2)

Default zero point

If the C-word is not programmed the following default settings are used:

For a workpiece blank the G199 coordinates are related to the program zero point W (C2).

For machine parts the G199 coordinates are related to the machine zero point M₀ (C1).

Coordinates of contour position

The coordinates of the contour position must be absolute, Cartesian coordinates. Polar coordinates are not allowed. These coordinates are related to the point defined with the C-word

Defining the position with E-parameters is not possible. Also defined points cannot be used.

The position of the contour is related to the program zero point W.

Type of contour description (B)

A contour description can be used for four purposes:

- 1 to define a blank contour (uncut part) (B1=)
- 2 to define machine parts, such as clamping devices, etc. (B2=)
- 3 to draw a contour of a machined work piece. (B3=)
4. to draw geometry elements (Line or circle) (B4=).

Workpiece blank (B1)

The outer and inner contour of a workpiece blank, thus the form of the uncut material, can be described with the graphic functions. The cutting of this material to the required shape can be simulated on the control.

Refer to the functions G197/G198 for details about describing a contour.

Machine parts (B2)

The contours of machine parts, such as clamping devices, can also be described by using the G199 function. This allows possible collisions between the cutting tool and machine parts to be detected.

The details given for the description of workpiece contours also apply to machine parts, except the B-word in the G199 block, which must be B2.

The description of machine parts is usually at the beginning of the partprogram. However, the description blocks can also be placed later in the program, in a particular part of the machining cycle.

A number of shapes can be programmed in succession enabling a machine part to be made up of several layers.

Several machine parts can be defined in a part program.

With macros a library for machine parts can be built up.

Contours (B3)

The contours of a machining part (outside contour rectangular pocket) can be describes with the graphical function.

The contour is only visible in wire plot simulation.

For special information about the contour description see the functions G197/G198.

The special information of B1 and B2 is also valid for the B3.

The contours can be describes all over in the program. More contours are possible.

There are no differences between G198 and G197 contours.

The contours are be drafted in the colour Cyan.

Geometric elements (linie or circle) (B4)

Geometric elements (line or circle) can be drawn with the G198-functionality. Spatial geometric elements will be projected on the active plane of the graphic function. So a complete drawing can be made.

With G199 B4 must defined that geometric elements will be drawn

The contour is only visible during simulation in wireplot graphic.

For description of the geometric elements refer to function G198.

Tool image

A tool image can be assigned to the tool with the aid of the G-word in the tool memory. The required image can be selected from a set of available tool images and is used by the CNC system to accurately simulate the machining.

Refer to the user manual for selecting the tool image.

Graphic functions and macros

The function for starting the contour description (G199), the contour description itself (G198 and / or G197) and the function for ending it (G196) must be in the same partprogram or subprogram (macro).

E-parameter can be used with the function G199, but parameter definition is not allowed in the G199 block itself.

Another blank contour description

In a part program more than one contour for a blank workpiece can be described. Each contour must be defined with the functions G199, G198, G197 and G196. A new graphic window (G195) must be defined too.

Only one blank contour is shown on the display. As soon as another description is encountered, the previous one is deleted and the other model shown. The tool movements are shown in the displayed model.

Plane selection (G17, G18, G19)

If a function for plane selection (G17, G18, G19) is encountered, the displayed program is deleted. After the G-function for plane selection another contour description can be given. The programmed movements are displayed in this model.

Only the movements in the last programmed plane are shown.

Cylinder interpolation

Movements programmed in the plane of the cylinder (G182) cannot be visualized on the display of the control.

Restrictions

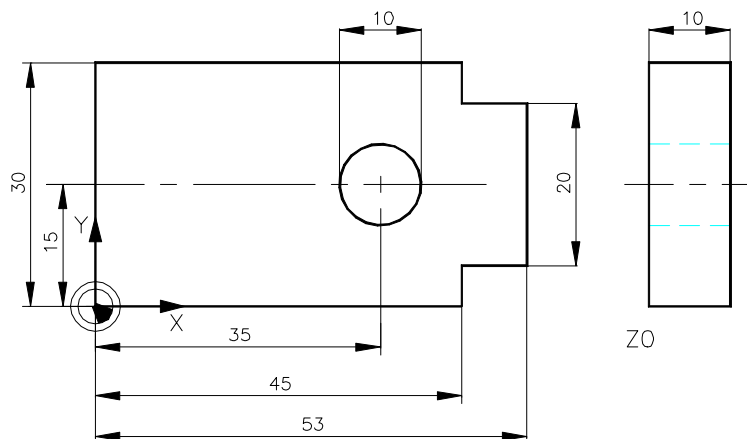
1. A G198 function must immediately follow the G199 block.
2. The G199 function cannot be used in MDI or TEACH-IN (PLAYBACK) mode.
3. A contour description should not be repeated with a G14 at another place. In this case the control repeats the model and the operations executed in this model before the G14 and does not stop properly. It is advised to write the blocks again in the program or to put the contour description in a macro.
4. A contour description is only allowed in the basic coordinate system, thus G180 active.

Operation with a contour description

1. A described contour between a G199 and a G196 is treated as one block. This means that at SINGLE BLOCK the complete contour is executed in one step.
2. A MANUAL BLOCK SEARCH to a block between a G199 and a G196 is not possible and results in an error message.
3. INTERVENTION during the drawing of the contour is not possible. The INTERVENTION is executed once the complete contour is drawn.

Example

Example 1 Defining a blank contour with clamping devices



Each clamp is described in a separate macro. With two parameters the start point of the clamp contour is programmed:

- E1 X-coordinate of the contour start point related to the program zero point.
 E2 Y-coordinate of the contour start point related to the program zero point.

The contour itself is programmed with fixed dimensions.

If this clamp is used in different part programs, the two parameters must be set at the macro call and then the clamp can be used for graphical purposes.

Macro for the left clamp

```

N1991
N1 G92 X=E1 Y=E2
N2 G199 X0 Y0 Z0 B2 C2
N3 G198 X0 Y0 Z0 D10
N4 G1 X45
N5 Y5
N6 X53
N7 Y25
N8 X45
N9 Y30
N10 X0
N11 Y0
N12 G197 X30 Y15 D-10
N13 G2 I35 J15
N14 G196
N15 G92 X=-E1 Y=-E2

```

Macro for the right clamp

```

N1992
N1 G92 X=E1 Y=E2

N2 G199 X0 Y0 Z0 B2 C2

N3 G198 X0 Y0 Z0 D10

N4 G1 X-45

N5 Y-5
N6 X-48
N7 Y-25
N8 X-45
N9 Y-30
N10 X0
N11 Y0
N12 G197 X-30 Y-15 D-10

N13 G2 I-35 J-15

N14 G196
N15 G92 X=-E1 Y=-E2

```

A zero point shift to let the start point of the clamp coincide with the program zero point.

Start the contour description section of the clamp. Its zero point is indicated in the drawing. Coordinates related to the program zero point are used.

Start of the outer contour description of the clamp. Offset values relative to point stated in G199-block.

Describe the outer contour. Coordinates are related to the point defined in the G199- block. The depth D is measured from the surface (Z0 in G199).

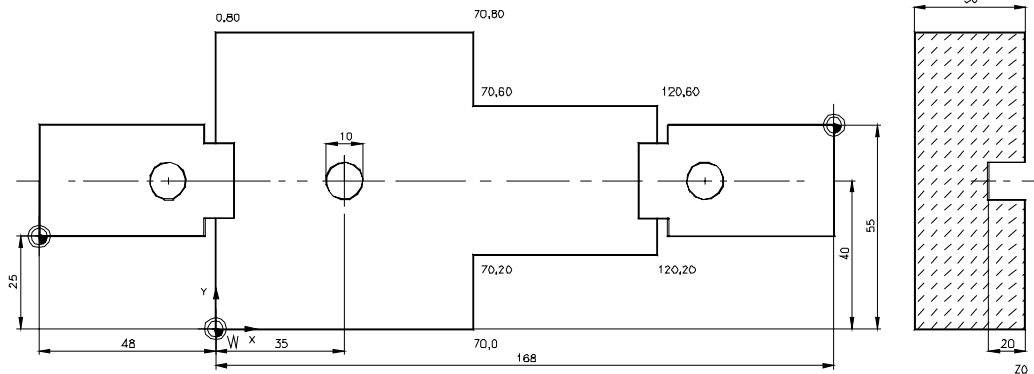
Start of the inner contour description. Offset values relative to point stated in G199- block. The depth D is measured from the surface (Z0 in G199).

Describe the inner contour to be a hole. Centre point coordinates are related to the point defined in G199-block.

End of the description section of the clamp.

Restore the program zero point.

Example 2 The graphic section of the part program:



N199000

N1 G17

N2 G54

N3 S1200 T1 M6

N4 G195 X-20 Y-20 Z-60 I160 J110 K70

N5 G199 X0 Y0 Z0 B1 C2

Set the main plane to be the XY-plane

Set the zero point

Load the tool and set the spindle speed

Set the graphic window to define the 3D space

Start the contour description section of the workpiece blank. The start point of the contour coincides with the program zero point.

N6 G198 X0 Y0 D-50

Start of the outer contour description. Offset values relative to point stated in G199- block.

N7 G1 X70

Describe the outer contour. Coordinates are related to the point defined in the G199- block. The depth D is measured from the surface (Z0 in G199).

N8 Y20

N9 X120

N10 Y60

N11 X70

N12 Y80

N13 X0

N14 Y0

N15 G197 X30 Y40 D-20

Start of the inner contour description. Offset values relative to point stated in G199- block. The depth D is measured from the surface (Z0 in G199).

N16 G2 I35 J40

Describe the inner contour to be a hole. Centre point coordinates are related to the point defined in G199-block.

N17 G196

End of the description section of the blank contour.

N18 G22 N=1991 E1=-48 E2=25

Call the macro for the left clamp

N19 G22 N=1992 E1=168 E2=55

Call the macro for the right clamp

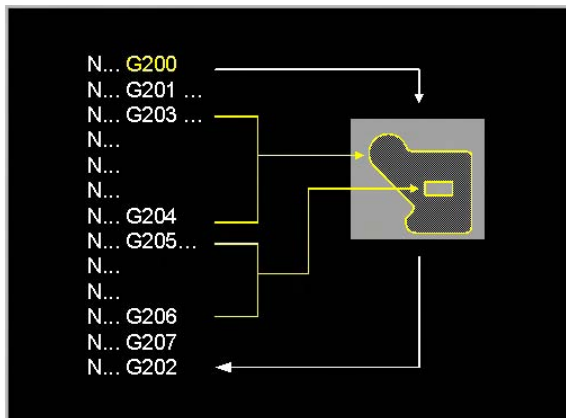
5.75 G200---G208 Pocket Cycle

5.75.1 G200 Begin pocket cycle

The Universal Pocket Cycle allows easier and quicker production of CNC programs, which control the milling of rectangular and circular pockets. 'Islands' of uncut material within the pockets can also be produced.

Format

G200



5.75.1.1 Introduction universal pocket cycle

The Universal Pocket Cycle (UPC) allows easier and quicker production of CNC programs, which control the milling of rectangular and circular pockets. 'Islands' of uncut material within the pockets can also be produced.

The CNC can calculate the minimum number of starting points to produce the required pocket contours in the shortest time. Tool movements are based on using the contour-parallel method, which is the most effective technique for machining pockets.

A 'start point' macro can be generated and used to pre-drill the cutter start position(s) in the workpiece. The minimum number of start points required to machine the pocket are calculated, keeping machining times to a minimum.

The programmer only needs to state the program numbers of the required UPC macros (subprograms), together with the machining parameters such as the feedrate, clearance distance, cutter radius etc., and the start point and dimensions of the pocket contour.

The CNC uses the above data to calculate the start points and coordinates of the toolpaths, which will be parallel to the contour's sides. These calculations are performed before the pockets are milled. A UPC macro is calculated only once; the contents of this type of macro will be re-used whenever the macro is reactivated in the CNC program.

The macro is calculated only once. The macro contents are reused every time the macro is called in the CNC program. If changes have been made (e.g. different cutter), the macro is recalculated.

The tool movements proceed to a contour-parallel process, which enables milling pockets to be machined most economically.

A 'finishing' cutting path can be incorporated in a machining cycle as a separate UPC macro.

This function must be programmed before the universal pocket cycles that have to be calculated, and commands that:

- if the calculations have not yet been made, the coordinates of the cutter paths must be calculated
- the cutter paths will be included in a macro generated by the CNC; the number (N1=...) of this machining macro is programmed in the G201-block
- if necessary (stated in a G201-block, by N2=...), a second macro will be generated for drilling the starting points
- if necessary (stated by the G203- or G205-function), the macros (N1=...) for finishing the contours will be generated.

All operational conditions such as the machining plane, zero point shifts and tool offsets, must be active before the G200-function is executed.

Point definitions (G78), which will be used for the pocket contour definition, must be defined before the G200-block.

All universal pocket cycles, which have been programmed between a G200- block and G202 or M30, will be calculated (if necessary).

A G200-block may be included in a macro, however the pocket will only be searched for in macros, which are nested deeper.

The CNC calculates the UPC macros before executing the CNC partprogram. Therefore, any blocks between G200 and G201 will initially be ignored. After the macros have been generated, these blocks will then be executed.

The generated macros are in the macro memory not visible for the operator. If the macro must be used in another program, the macro number must be entered in the macro memory. Then the macro is visible in the macro memory and the macro can read in and out.

For reasons of memory space the pocket contour and associated islands may not exceed 50 sides. 15KB of additional memory capacity will be necessary to allow approximately 50 sides to be stored. A circular movement that is greater than, or equal to, 180° will be treated as if it had two sides because the CNC system automatically divides this type of movement into two equal parts.

The machining plane (G17/G18/G19) must be selected before the G200-function or after a G202-function, is executed.

If the coordinates of the defined points are changed after the pocket has been calculated, and if the pocket must be calculated again, the macros generated during the calculation of the second pocket will be stored. The macros of the first calculation will be destroyed.

For calculating the macros, characteristics and quantities such as programmed points, scaling, rotations, mirror image etc. will be used the way they are active with the G200-block concerned.

If errors are generated in a pocket cycle program or graphics shows material left, the programmer is advised to make alterations to the program in order to prevent those errors. Proposals are changing the overlap percentage, change-programming sequence, change entry point, split up program, and define pseudo-island of 1 micron.

If a calculation error occurs during execution of a pocket cycle (0170, 0176) please check the pocket contour. If the contour seems ok check the tool radius size with respect to contour and islands, and choose a tool with a smaller radius when the originally programmed tool is comparatively large.

If in pocket cycles the tool radius is relatively large compared to the pocket or its islands, some problems may occur:

- G201 I Certain I-values may lead to material left or damage of islands programmed.
- Errors P163, 0170 or 0176 could be set.

The only cure in above cases is to choose a smaller tool for machining this pocket cycle concerned.

5.75.1.2 Part program structure

The example below shows a simple program, which uses a macro, which specifies a pocket cycle.

```

N99999 G54
N1 G17
N2 \
: > Normal machining
N96 /
N97 G200                Begin pocket cycle
N98 G81                Drill cycle description
N99 G22 N=..           Start point drilling
N100 G201 N1=.. N2=..   Starting to mill pocket cycle
N101 G203 N1=..        Begin pocket contour description
N102 \
: > Pocket contour description
N109 /
or
N101 G208 N1=..        Pocket contour description of a regular quadrangle.
N110 G204               End pocket contour description
N111 G205 N1=..        Start island contour description
N112 \
: > Island 1 contour description
N118 /
N119 G206               End island contour description
N120 G205 N1=..        Start island contour description
N121 \
: > Island 2 contour description
N129 /
N130 G206               End island contour description
N131 G207 N=..        Island contour description 3 is a macro
N132 G202               End of pocket cycle

N350 G22 N=..           Pocket contour finishing
N351 G22 N=..           Island 1 finishing
N352 G22 N=..           Island 2 finishing
N353 G22 N=..           Island 3 finishing

```

5.75.1.3 Translation, rotation and mirror image of a pocket

A pocket cycle can be described using a datum point which is different from the datum point used during program execution; this may be necessary for either arithmetical reasons, or if the pocket is included in a macro.

A different datum point is established by programming a datum point shift and/or axis rotation before the G201-block.

The datum point shift and/or axis rotation are programmed with the standard function G92 or G93. The axis transformation will then be performed on the generated machining macro.

When the pocket has been cut the programmer must ensure that the program datum point is reset at the correct location. Similarly, when using the macro for the starting points and macros for finishing the pocket contour, the programmer must ensure that the correct datum point is used.

A pocket may be used again within the same part program by programming a new start position and orientation.

Example

```

N9001
:
N90 G200
N100 G201 N1=9999
N110 G203
:
: \
: > Description of the pocket contour
: /
N200 G204

N205 G205
:
: \
: > Description of an island contour
: /
N206 G206
N300 G202
:
N400 G92 X.. Y.. Z.. B4=..
N410 G22 N=9999
:
N500 G73 X-1
N510 G22 N=9999

```

Explanation:

In block N400 the datum point is shifted and the pocket rotated (B4=..), so that the starting point of the pocket is correctly positioned. In block N410 the machining macro is called. The pocket will be cut again but at a different location.

In this example (with G17 active), mirroring about the Y-axis is programmed in block N300 and the machining macro executed in block N310 as mirror image. The disadvantage is, that one pocket is machined in a backward direction and the other in a forward direction. When this is not possible for technological reasons, the mirror image function cannot be used. The mirrored pocket must then be programmed again.

5.75.1.4 Same pocket in another program

If a pocket (and associated islands) occurs in different programs, the complete pocket may be written in a macro (subprogram); this macro will then be called at that point in the partprogram where the pocket must be cut.

The functions G201, G202, G203/G204, and G205/G206 have to be stated in the macro.

Programming will be:

```
N9001 G201 Y.. Z.. B.. N1= N2=
N1 G203
:           \
:           >   Pocket contour
:           /
N8 G204
N9 G205
:           \
:           >   Contour of island 1
:           /
N13 G206
N14 G205
:           \
:           >   Contour of island 2
:           /
N18 G206
N19 G202
```

Represents a macro of a pocket, which has islands.

A partprogram, which uses this macro, could look like:

```
N9999
N1 G200
:
N50 G22 N=9001
:
```

A datum point shift prior to the macro call positions the pocket in the correct location.

Remark By including the G201-block in a macro, the nesting level of the machining macro will be increased by one. Macros cannot be nested more than 8 times.

5.75.1.5 Operating section

Macros for pockets, starting point and finishing macros

Generation

When the control encounters a G200-block in a program, it searches for the pocket functions (G201-block + associated contour description + G202-block). Upon finding a G201-block (in the partprogram or possibly via a macro call) and if no macro carrying the programmed macro number (N1=.. in the G201-block) exists, a pocket macro will be generated by the control. (The data, which determine the pocket macro, viz. the parameters Y, Z, B, R, I, K in the G201-block and the contour description, will be stored with the pocket macro).

If a macro with the programmed pocket macro number exists already, the associated data will be compared with the programmed data. If they do not match, a new pocket macro will be generated.

A new starting points macro is generated if:

a starting points macro number is programmed (N2=.. in the G201-block) and a new pocket macro is generated or if no macro with the programmed starting points macro number exists.

Finishing macros are generated if:

a finishing macro number is programmed (N1=.. in the G203- or G205-block concerned) and a new pocket macro is generated or if no macro with the programmed finishing macro number exists.

Removal

After being generated the macros are "locked". In order for them to be removed they must first be "unlocked". Removal may be necessary, for example, if a G201-block is being removed from a program or if an 'N1=' or 'N2=' address in a G201-block or an 'N1=' address in a G210-block is being changed.

The macros will be automatically removed if a program is started in which the pocket definition has been changed.

In V330 are for the operator, the generated macros in the macro memory no longer visible. For using a macro in another program, the macro number must be given in the macro memory. After that the macro will be visible in the macro memory. It is possible to read-in and read-out the macro program.

Sign of life

The Universal pocket cycle software is designed to stop all calculations upon detection of an error. To indicate that the CNC is busy a rotating indicator, with the word "CLOCK", is displayed on the screen. While the indicator is rotating the CNC is busy calculating the pocket cycle.

Teach-in

The use of a pocket G-function (G200 to G208) in TEACH-IN MDI and TEACH-IN/PLAYBACK is not permitted.

Editing

During editing, static or dynamic programming support may be used.

A (new) program may be edited while a program with a pocket section is running.

block delete

If a G201-block contains a "/" character and "block delete" is activated, then:

G200 will be executed in the normal way, i.e. macros will be generated if necessary. The G201-block, the contour description and the G202-block will be skipped during the program execution, i.e. program execution will continue after the G202-block.

Block search

Searching a G200-block causes pocket macros to be generated if they are not yet present.

During program execution (after the G200-block has been executed), G201 is taken as a macro call (G22) to the pocket macro. However, after executing the pocket, a jump is done to the first block after the terminating G202-block.

Searching for a block in a pocket macro is performed in the same way as searching for a block in a macro called via G22.

Note: As G201 is taken as a macro call during program execution, the nesting level (max. 8 levels) should be taken into account when programming G201.

Intervention

Intervention during the execution of a G200-block is possible. However, macros have to be recalculated after an intervention and this could take a long time if a complex contour is required.

Execution can be resumed via "start". After intervention, it is not possible to enter the edit mode without a "clear control".

Intervention during the execution of a pocket macro is dealt with in the same way as intervention in a macro called with G22.

Incomplete programming

Normally, the following G-functions are programmed for a pocket cycle: G200, G201, G202, G203/G204, G205/G206, G207 and if necessary, G208.

G201 and G202, G203/G204 and G205/G206, are required to appear as a combination in the same program or the same macro, otherwise an error message is issued.

If a G203/G204 or G205/G206 combination appears (possibly via a macro call G22) without the associated G201/G202 combination, an error message is issued.

A G207 should always appear after a G205/G206 combination, with both G205/G206 and G207 belonging to the same G201-block, otherwise an error message is issued.

If a G201/G202 combination is programmed without contour descriptions, an error message is given.

If G200 is programmed without G201/G202 contour descriptions, the program is executed in the normal way (without pocket cycle).

If G201/G202 contour descriptions have been programmed without G200, no macros are generated and G201 is taken as G22,

Operation mode change

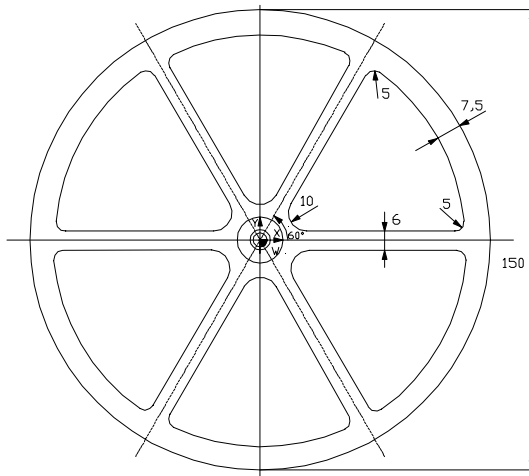
An operational mode change can only occur after a G200-block has been completely executed.

5.75.1.6 Error messages

- #5 Memory pool exhausted
- O170* Pocket-cycle calculation error (Operating error).
Solution: Decrease number of parameters (MC 83).
- O176* Module xxx, number xxx.
Note: The 0170 and 0176 messages are displayed together. Solving this problem can be achieved by using one of three methods:
- Change the tool radius.
 - Change the overlap I-word value
 - Change the amount of stock to be left for the finishing cycle
- P07 Programmed data is out of range
In a block with a circular (G2/G3) side programmed with a radius value. When the programmed radius is two times greater than the distance between the start and end point. This can occur on a side of 180°.
The error is removed by changing the start or end point.
- P35 End point not on circle.
On a block with a circular (G2/G3) side programmed with centre point or a block with a circular (G2/G3) side programmed with geometry. The value of the programmed centre point is greater than the value of the Circular end point window (calculated centre point between start and end point) specified by machine constant 712.
The error can be removed by changing the centre point or the start/end point.
- P75 Circle without centre point
An arc has been programmed which has the same start and end point.
The error is removed by deleting the program block or by programming an arc, which has a separate start and end point.
- P140 Invalid G207 nesting
- P141 Too many sides programmed
- P142 Too many contours programmed
- P143 Invalid 'G' in pocket-cycle mode
- P144 Invalid contour description
This error is caused by one of the following reasons:
- The G208-function has been omitted
 - The G208-function specifies rounding (radii) which are too large
 - The G208-function block contains X=0 or Y=0
 - The G208-function block contains B1 > 180°
 - The G208-function block contains an I or R-word
- P145 Invalid start point for approach.
A finishing cutting path cannot be generated from the current description, another start point must be chosen.
- P146 No G202 defined
- P147 Memory manager error
- P148 Floating point error
- P150 Tool not found
- P160 Pocket macro generation error
- P161 Finishing macro generation error
- P162 Macro start point generation error
Note: Errors P160, P161 and P162 are produced by:
- a full CNC memory
 - Too many start points having to be calculated
- P170 Contour xx not closed
- P171 Contour xx has more inner areas
- P172 Contour xx intersects contour xx
- P173 Contour xx enclosed by contour xx
- P174 Contour xx is outside the pocket

Example

Example1 Rotated pocket



N3620511 (WHEEL AS POCKET)
 N1 G17
 N2 G54
 N3 G195 X-90 Y-90 Z0 I180 J180 K-10
 N4 G99 X-85 Y-85 Z0 I170 J170 K-10
 N5 G200
 N6 T31 M6 (DRILL RADIUS 4. mm)
 N7 G81 Y1 Z-5 F100 S100 M3
 N8 G22 N=3620501
 N9 G92 B4=60
 N10 G14 J5 N1=8 N2=9
 N11 G93 B4=0
 N12 T04 M6 (ROUGHING MILL RADIUS 3. mm)
 N13 S1500 M3
 N14 G201 Y0.1 Z-5 B1 I50 F1000 N1=3620500 N2=3620501 F2=500
 N15 G203 X37.5 Y3 Z0 N1=3620502
 N16 G64
 N17 G1 X1=0 Y1=3 B1=0 J1=2
 N18 G3 R5
 N19 I0 J0 R67.5 J1=1
 N20 R5
 N21 G1 X1=0 Y1=0 B1=-120 I1=-3
 N22 G3 R10
 N23 G1 X37.5 Y3 B1=0
 N24 G63
 N25 G204
 N26 G202
 N27 G92 B4=60
 N28 G14 J5 N1=14 N2=27
 N29 G93 B4=0
 N30 T3 M6 (FINISHING MILL RADIUS 2.5 mm)
 N31 S1800 M3
 N32 G22 N=3620502
 N33 G92 B4=60
 N34 G14 J5 N1=32 N2=33
 N35 G0 Z100 M30

Explanation:

The CNC processes the blocks N3 and N4, which define a graphic simulation of the program's operation.

The UPC macros are calculated first (blocks N5 and N14 to N26 are executed). The following macros are created:

- Macro No. 3620500 for the machining-cycle;
- Macro No. 3620501 for the starting-points;
- Macro No. 3620502 for the finishing-cycle.

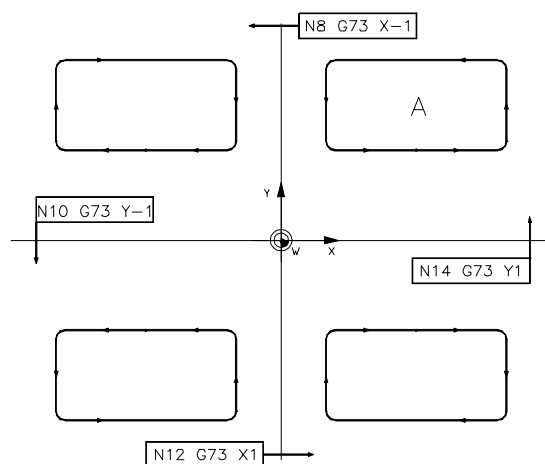
Tool 31 is selected and a drilling cycle defined by the function G81. CNC control is transferred to UPC macro no. 3620501 by the use of E-parameter no.1 and the G22-function. The first starting points are drilled. Block N9 commands the macro's coordinates to be rotated 60° in a counter clockwise direction.

Tool 4 is selected. Block N14 commands that the machining cycle (macro no. 3620500) is executed. When the machining cycle is finished, the coordinates of the macro 3620500 are rotated 60° in a counter-clockwise direction. Block N28 commands the machining cycle to be repeated and rotated five times (Note: N2=27 can be omitted from the block and the repeat instruction will still operate).

Tool 3 is selected. Block N32 assigns the value 3620502 to E-parameter no.1. The 'finishing' cycle is executed in the same manner as for the two previous UPC macros.

Block N35 commands the tool to move to Z100 and the partprogram to end.

Example 2 Mirrored pocket



A: Programmed pocket cycle

N3620513 (MIRROR IMAGE OF A POCKET)

N1 G17

N2 G54

N3 G195 X-150 Y-110 Z0 I300 J220 K-10

N4 G99 X-145 Y-105 Z0 I290 J210 K-10

N5 G200

N6 T31 M67 (DRILL RADIUS 4. mm)

N7 G81 Y1 Z-5 F500 S1000 M3

N8 E1=3620501

N9 G22 N=E1

```

N10 G73 X-1
N11 G14 N1=9
N12 G73 Y-1
N13 G14 N1=9
N14 G73 X1
N15 G14 N1=9
N16 G73 Y1
N17 T4 M6 (ROUGHING MILL RADIUS 3. mm)
N18 S1800 M3
N19 G201 Y0.1 Z-5 B1 I60 N1=3620500 N2=3620501 F1000 F2=500
N30 G203 X75 Y50 Z0 N1=3620502
N21 G1 X120
N22 G3 X125 Y55 R5
N23 G1 Y95
N24 G3 X120 Y100 R5
N25 G1 X30
N26 G3 X25 Y95 R5
N27 G1 Y55
N28 G3 X30 Y50 R5
N29 G1 X75 Y50
N30 G204
N31 G202
N32 E1=3620500
N33 G14 N1=10 N2=16
N34 T3 M6 (FINISHING MILL RADIUS 2.5 mm)
N35 S2000 M3
N36 E1=3620502
N37 G14 N1=9 N2=16
N38 G0 Z100 M30

```

Explanation:

The CNC first processes blocks N3 and N4, which defined a graphics simulation of the partprograms operation.

The UPC macros are calculated first (blocks N5 and N19... N31 are executed). The following macros are created:

- Macro No. 3620500 for the machining-cycle;
- Macro No. 3620501 for the starting-points;
- Macro No. 3620502 for the finishing-cycle.

Tool 31 is selected and a drilling cycle defined by the function G81. CNC control is transferred to UPC macro no. 3620501 by the use of E-parameter no.1 and the G22-function. The first starting points are drilled.

Blocks N10 to N16 command the coordinates of the starting-points macro to be mirrored about the X and Y-axes and for the macro to be executed after each mirroring. The G22-function transfers CNC control to the macro and the G14-function commands the block N9 to be repeated once.

Tool 4 is selected after the pocket has been executed four times. Block N32 assigns the value of 3620500 to E-parameter no.1; block N33 commands that blocks N10 to N16 are repeated. The 'machining' macro is therefore executed once and then mirrored and repeated in the same manner as for the 'starting-points' macro.

Tool 3 is selected. Block N36 assigns the value 3620502 to E-parameter no.1. The 'finishing' cycle is executed in the same manner as for the two previous UPC macros.

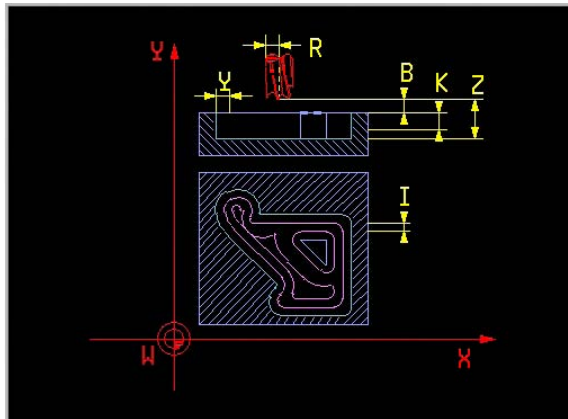
Block N38 commands the tool to move to Z100 and the partprogram to end.

5.76 G201 Start contour pocket cycle

Entering technological data to calculate the pocket cycle. Pocket milling starts with this block.

Format

G201 N1=.. Z.. {N2=..} {Y..} {B..} {R..} {I..} {J..} {K..} {F..} {F2..}



G Start contour pocket cycle
 Y Stock removal
 Z Total pocket depth
 B Clearance
 I Cutting width mill in %
 J 1=climb, -1=conventional
 K Cutting depth
 R Tool radius for calculation
 N1= Milling macro number
 N2= Startpoint macro number

N1= Number of the machining macro. This number has to be programmed.

N2= Number of the starting points macro.

Y Machining allowance = amount of material required to be left on the contour for finishing The Y-word carries no sign. If Y has not been programmed, Y=0 will be used as a default.

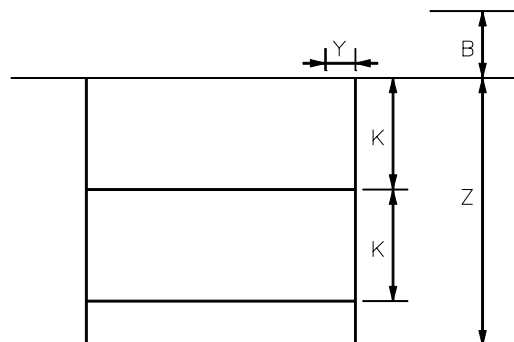
Z The total depth of the pocket. The depth is measured from the coordinate of the tool axis from the G203-block (which is equal to the position of the upper surface of the pocket). A negative Z-word value is the depth in the negative direction of the tool axis. A positive Z-word value is the depth in the positive direction of the tool axis. The Z-word must always be present in a block, which contains the G201-function.

These words are independent of the selected machining plane.

B Clearance distance above the pocket. This distance is measured from the coordinate of the tool axis specified with the G203-function.

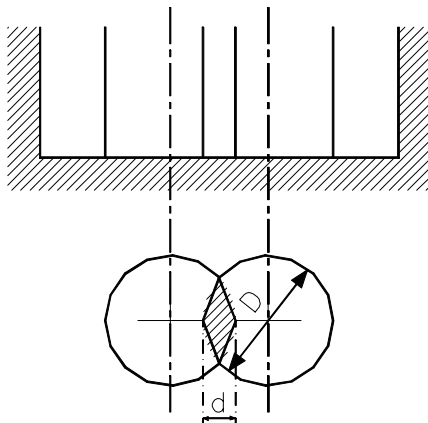
Positive B-word value is the clearance distance in the positive direction of the tool axis; a negative value is the clearance distance in the negative direction. The sign of the B-word must always be opposite to the sign of the Z-word. If a B-word is not been programmed, B=0 will be used as a default.

The B-word is used to define the position on the tool axis where the feed movement begins for entering the pocket at each starting point. At the end of the machining operation the tool is retracted above the pocket surface, at a distance stated by the B- Word.



- R The cutter radius to be used for calculating the cutter paths. The final cutter radius, which is actually used, can be different. The R-word carries no sign and must always be programmed.
- I The amount of overlap between cutting passes. The overlap distance is specified as a percentage of the tool's diameter e.g. I75= 75% of the diameter. I lies between 1% and 100%.

The I-word carries no sign. If the I-word is not programmed, the value of MC720 will be taken.



D= Cylinder cutter diameter
d= overlap distance
 $I = 100 (1 - (d/D))$

- J The direction of movement for rough milling.
J1 (default value): counter-clockwise direction
J-1: clockwise direction.
- K The cutting depth per path. The K-word carries no sign. The last feed-in distance may be smaller than the value of K for the final cutting pass. If the value of K is larger than that of Z, or K-word is not stated, the Z-value will be used i.e. the value of $K = |Z|$.
- F The feed during milling.
If F is not programmed, the last programmed feed will be used.
- F2= The feed for moving to a next machining plane. If the holes have been pre-drilled a great value can be taken. If F2= is not programmed, the last programmed feed will be taken.

The functions G90, G40 and G63 will be automatically activated when the G201-function is executed. G90 is required to be active because the generated macros use absolute dimensions and the first position of the pocket contour definition (G203-block) must be absolute.

5.76.1 Usage of the generated macros

Starting point macro

The control is capable of generating a starting points macro. The starting points macro contains the points where the cutting tool enters the material for "roughing out" the pocket (these points are calculated by the control). The programmer can command a hole to be drilled in these positions, so that the cutting tool need not cut in the direction of the tool axis.

The macro generated by the CNC will be similar to that given below:

N (is N2- word of the G201-block)

N1 G90	Absolute programming
N2 G79 X.. Y.. Z..	Activate a predefined drilling cycle
N3 G79 X.. Y.. Z..	
N4 G79 X.. Y.. Z..	Depending of number of drilling holes

The position in the machining plane XY (G17), XZ (G18) or YZ (G19) is given in the coordinates of the axes system in which the pocket has been described. The tool axis Z (G17), Y (G18) or X (G19) is given by the G203-block of the pocket contour.

After execution of the starting-points macro, G90 becomes active. The starting-points macro is generated simultaneously with the machining macro.

A partprogram could be as follows:

```

N9900
N1 G200
:
N90 T1 M6 (Drill)
N100 G81 X.. Y.. Z.. B.. F.. S.. M3
N110 G22 N=9902
:
N200 G201 Z.. N1=9901 N2=9902
: \
: > Description of the pocket (including islands)
: /
N300 G202

```

The cycle for pre-drilling the starting points is defined in block N100. The starting points macro is called in block N110. The cycle from block N100 is executed on the starting points.

Machining macro

The machining macro is generated by the control and includes all the movements necessary for roughing out a pocket. This macro (subprogram) will be called when the G201-function is executed. When the G202-function terminates the macro, the G40 and G90-functions will always become active automatically.

The example below shows a simple program, which uses a macro, which specifies a pocket cycle.

```

N (is N1-word in G201-block)
N1 G40          No radius correction
N2 G90          Absolute coordinates are used
N3 G0 X.. Y.. Z.. Tool moves to start point
N4 G91          Incremental coordinates are used
N5 G1 Z.. F..   Tool is fed to depth
N6 G90          Absolute coordinates are used
N7 F..          Feed
: \
: > Pocket milled
: /
N99
N100 G91
N101 G0 Z..     Tool is retracted
N102 G90        Absolute coordinates used again
N103 G0 X.. Y.. Tool moves back to starting point
N104 G14 N1=4 N2=103 J.. Pocket milling is repeated
N105 G91        Incremental coordinates are used
N106 G1 Z.. F..
N107 G14 N1=6 N2=99 Final cutting pass is performed
N108 G0 Z..     Tool retracts out of work piece

```

5.76.2 Macro for finishing a pocket contour

The macro for finishing the pocket contour is generated by the control. The macro includes all the movements including a circular or linear feed-in and feed-out, necessary for finishing the pocket sides.

Feed-in point

The feed-in point S is determined by the CNC. This point is always near the start position of the pocket and at an equivalent distance (d) to both first and last sides.

The CNC will calculate different feed-in points depending on which contour side the tool has to first move to. The programmer therefore has to determine the correct tool size and the order in which the contour sides are stated, to prevent the tool colliding with the workpiece.

The CNC uses the formula below to calculate point S.

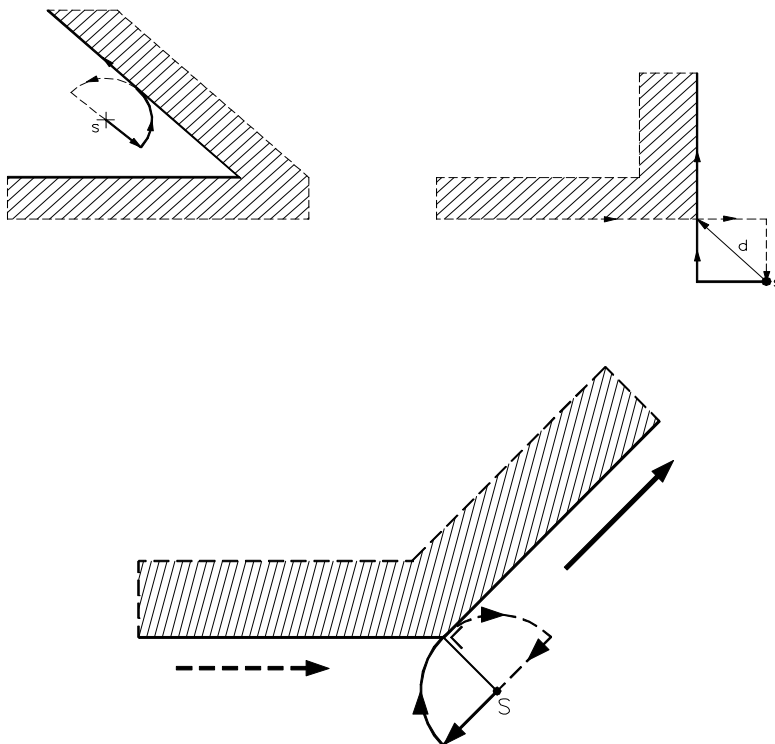
Distance = factor * radius : 100

factor = given per machine constant (MC719) (between 101..200).

radius = the value of the R-word from the G201-block, or the active radius during G201, if not programmed

Within the circle, with S as centre and distance as radius no contour element of an island or pocket is allowed.

If it is not possible to calculate point S, the CNC generates an error code.

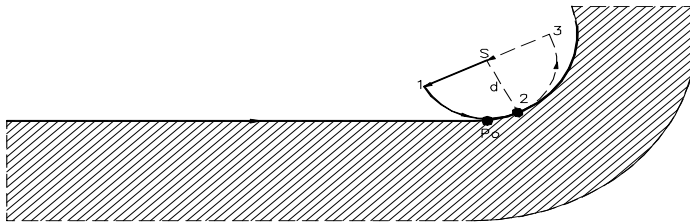


The feed-in movement is linear when the first and last movements are also linear and the angle between \Rightarrow 270 degrees (this angle is measured inside the pocket). A circular feed-in movement is used in all other instances.

If a different tool is used for the finishing cut the programmer must ensure that the tool does not collide with the contour.

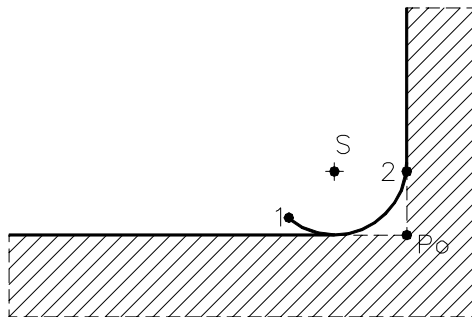
General format of the macro for a circular feed-in

The feed-in of the contour is effected via a quarter-circle with centre S and a radius equal to the distance between S and the contour.



$$\text{distance} = \text{factor} * \text{radius} : 100$$

A problem will arise when the start point of the feed-in circular movement is on the path of the final contour. The tool will then slightly touch the contour surface. To prevent this occurring the CNC automatically repositions the start point S.



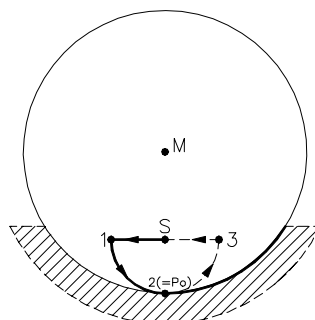
In this example the feed-in circle begins 120° before point 2.

A pocket wall can usually be machined in one step. If this is not possible, the programmer can finish the wall in several steps using datum point shifts.

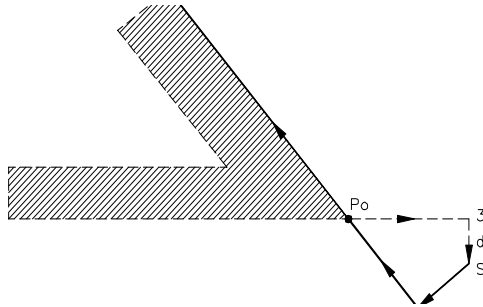
After completion of a macro, the functions G0, G40 and G90 become active.

Circular pocket contour

For a circular pocket contour the calculation of the feed-in point S must be changed.



In this case point S lies on the straight line M to Po, at a calculated distance from Po. A pocket contour will be entered at point Po (= point 2). The direction of rotation at feed-in and feed-out is the same as the programmed direction of rotation for the contour.

Linear feed-in movement

$$\text{distance} = \text{factor} * \text{radius} : 100$$

Note: After the macro is executed, the functions G0, G40 and G90 will be active.

5.76.3 Sequence of the macros on the machine

1. Starting points macro
This macro will run in the normal way without any restrictions.
2. Machining macro
This macro will run in the normal way as well, but some material may be left.

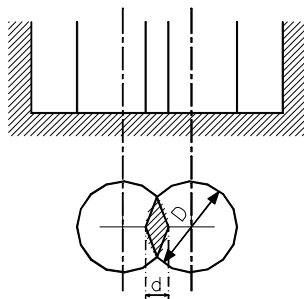
There are two possibilities:

- (i). During contour-parallel milling small areas may remain unmilled when the tool changes direction between two parallel paths; this situation can also occur at contour roundings:

The solutions to these two problems are:

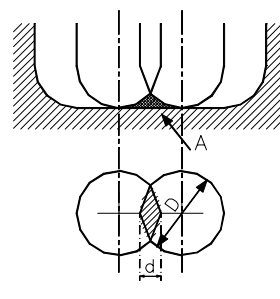
- a. For parallel movements - a bigger overlap between parallel movements.
 - b. For contour narrowing - a tool with a smaller diameter is used.
- (ii) Due to the shape of the cutting tool some material may be left on the bottom. Normally, the overlap-parameter (I-word from the G201-block) should ensure that no material is left between the paths. The greater the value of the I-address, the greater the risk that material will be left on the bottom.

A value of 50% will ensure that when a square-edge-milling tool is used no material will be left in the pocket's bottom surface. However, milling tools with curved edges require different I-values because of the different radii of the curved edges.



Cylinder cutter

D= cutter diameter
d= overlap distance
A= uncut material



Ball cutter

3. Macros for finishing

The radius of the active tool will be used for radius compensation. The user should check that the cutting tool could move without damaging the pocket side. The macros will operate in the normal way.

Notes and usage

The function G201 and the terminating function G202 must both be written in the same program or in the same macro.

Between G201 and G202 only the following functions are permitted:

G203, G204, G205, G206 with contour descriptions, G207 and G208

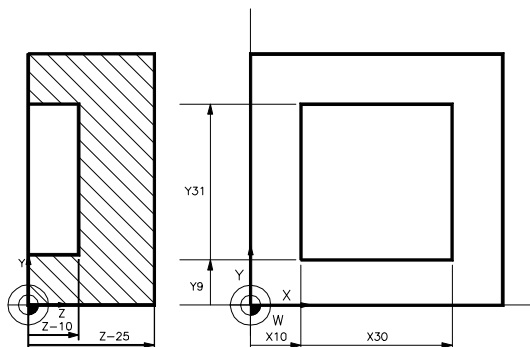
During description between G201 and G202 the following functions may not be used:

- Datum points shifts, notably (G92/G93) and stored zero offsets (G51 to G59) G54I[nr.]
- Axis rotation (G92, G93)
- Mirror image or scaling (G72, G73)
- Definition and activation of fixed cycles
- Measuring cycles
- Plane selection (G17, G18, G19)
- Point definition (G78)
- Helix interpolation
- Tool change (M06, M66, M67)
- Macro (G22) or program call (G23)
- Block and pattern repeat (G14)
- Conditional jump (G29)
- Chamfer or rounding (G11)

The use of E-parameters is not permitted for contour descriptions, or in the G201-block.

Examples

Note For the following examples it is assumed that G17 is active.

Example 1 Rectangle

Programming can be done in two ways:

N.. G201 N1=.. Z-10

N.. G203 X10 Y9 Z0 N1=9800

Absolute

N.. X40

N.. Y40

N.. X10

N.. Y9

N.. G204

N.. G202

incremental

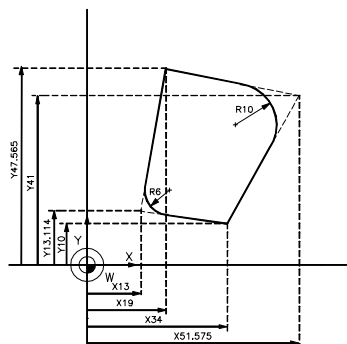
N.. G91

N.. X30

N.. Y31

N.. X-30

N.. Y-31

Example 2 Quadrangle with roundings

N.. G201 N1=.. Z-10

N.. G203 X34 Y10 Z0

N.. G64

N.. G1 X51.575 Y41

N.. G3 R10

N.. G1 X19 Y47.565

N.. G1 X1=13 Y1=13.114

N.. G3 R6

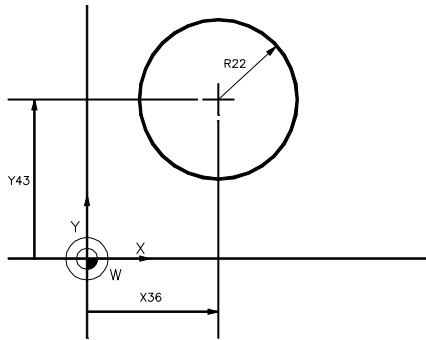
N.. G1 X34 Y10

N.. G63

N.. G204

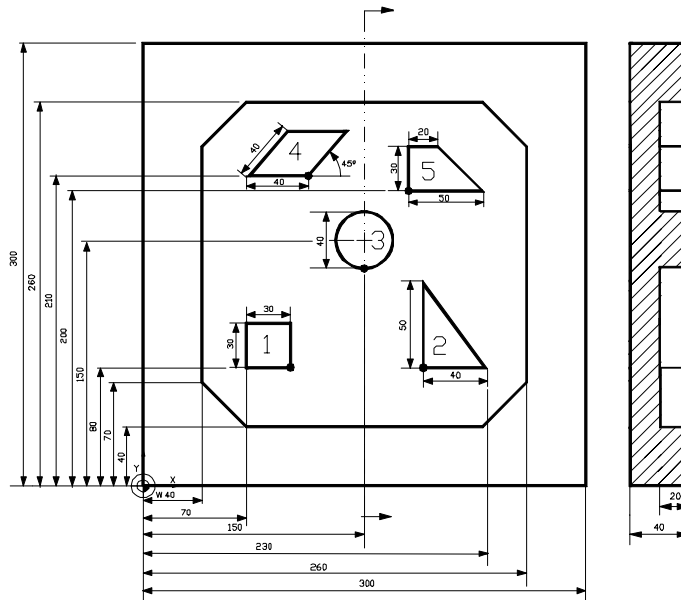
N.. G202

Example 3 Full circle



N.. G201 N1=.. Z-10
 N.. G203 X36 Y21 Z0
 N.. G3 I36 J43
 N.. G204
 N.. G202

Example 4 Pocket with islands



N9990

N1 G54
 N2 G17
 N3 G195 X-10 Y-10 Z10 I320 J320 K-60
 N4 G99 X0 Y0 Z0 I300 J300 K-40
 N5 G200
 N6 T2 M6 (predrilling start point, drill R10)
 N7 G81 Y2 Z-20 F200 S3000 M3
 N8 G22 N=9992
 N9 T3 M6 (clearing out the pocket, mill R8)
 N10 S2500 M3
N11 G201 Y0.1 Z-20 B2 I50 K8 F200 F2=1000 N1=9991 N2=9992
 :
 N37 G202

5.77 G202 End contour pocket cycle

End contour pocket cycle

Format

G202

Notes and Usage

The function G201 and the terminating function G202 must both be written in the same program or in the same macro.

Between G201 and G202 only the following functions are permitted:

G203, G204, G205, G206 with contour descriptions, G207 and G208

During description between G201 and G202, the following functions may not be used:

- Datum points shifts, notably (G92/G93) and stored zero offsets (G51 to G59) (G54I[nr.])
- Axis rotation (G92, G93)
- Mirror image or scaling (G72, G73)
- Definition and activation of fixed cycles
- Measuring cycles
- Plane selection (G17, G18, G19)
- Point definition (G78)
- Helix interpolation
- Tool change (M06, M66, M67)
- Macro (G22) or program call (G23)
- Block and pattern repeat (G14)
- Conditional jump (G29)
- Chamfer or rounding (G11)

Completion of the entire pocket description. After the pocket has been cut out, the partprogram will be resumed with the block following G202. Only the N-word is permitted in a G202-block.

By a G202-function the calculation for the universal pocket cycle will be stopped. After a new G200-function the calculation will be started again.

The functions G0, G40, G63 and G90 will be active after the G202-function is activated.

After the pocket description the program should be continued with an absolute position.

Example Pocket with islands**N9990**

N1 G54

N2 G17

N3 G195 X-10 Y-10 Z10 I320 J320 K-60

N4 G99 X0 Y0 Z0 I300 J300 K-40

N5 G200

N6 T2 M6 (predrilling start point, drill R10)

N7 G81 Y2 Z-20 F200 S3000 M3

N8 G22 N=9992

N9 T3 M6 (clearing out the pocket, mill R8)

N10 S2500 M3

N11 G201 Y0.1 Z-20 B2 I50 K8 F200 F2=100 N1=9991 N2=9992

N12 G203 X40 Y40 Z0 N1=9993

N13 G208 X220 Y220 I30 (pocket contour)

N14 G204

N15 G205 X100 Y80 N1=9994

N16 G208 X-30 Y30 J-1 (Island 1)

N17 G206

N18 G205 X190 Y80 N1=9995

N19 G91

N20 Y50 (Island 2)

N21 X40 Y-50

N22 G90

N23 G206

N24 G205 X150 Y130 N1=9996

N25 G2 I150 J150 (Island 3)

N26 G206

N27 G205 X110 Y210 N1=9997

N28 G208 X-40 Y40 J-1 B1=135 (Island 4)

N29 G206

N30 G205 X180 Y200 N1=9998

N31 G91

N32 Y30

N33 X20 (Island 5)

N34 X30 Y-30

N35 G90

N36 G206

N37 G202

N38 T4 M6 (clearing out the pocket, mill R8)

N39 F200 S2200 M3

N40 G22 N=9993

N41 G22 N=9994

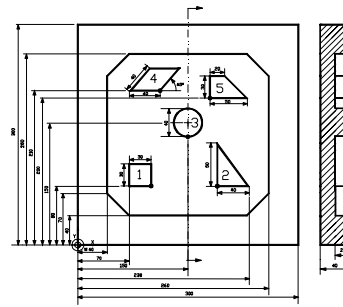
N42 G22 N=9995 (finishing)

N43 G22 N=9996

N44 G22 N=9997

N45 G22 N=9998

N46 M30

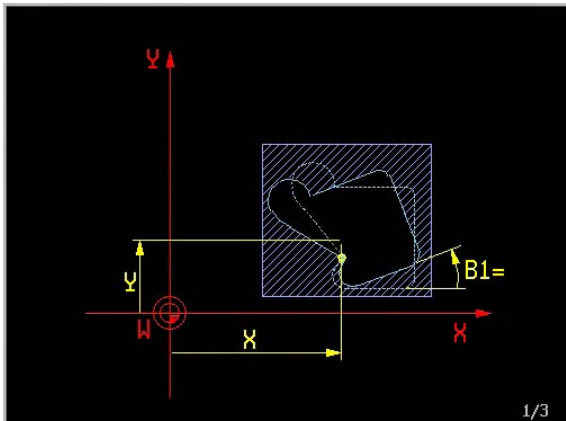


5.78 G203 Start pocket contour description

Start pocket contour description

Format

G203 X.. Y.. Z.. {B1=..} {N1=...}



G Start pocket contour description
 X Startpoint in X
 Y Startpoint in Y
 Z Startpoint in Z
 P Point definition number
 N1= Finishing macro number
 B1= Rotation angle of contour pocket
 B2= Start point polar angle
 L2= Start point polar length
 P1= Point definition number

- N1= Number of the macro for finishing the pocket contour. A finishing macro will not be generated if the word N1=.. is not programmed.
- B1= Rotation of the pocket contour around the point from the G203-block. Islands will not be rotated.

Notes and Usage

The description of the pocket contour is started by stating the position of the first point: the finishing cut of the contour also starts at this point.

The starting point can be defined by absolute Cartesian coordinates measured from the program zero point W. The starting point can also be programmed by using absolute polar coordinates (B2=... and L2=...). The position can also be programmed by a predefined (G78) point.

The position of the workpiece's upper surface is defined by coordinate values of the following tool axes: (G17) Z-axis, (G18) X-axis, (G19) and Y-axis.

A tool axis coordinate must always be present in a G203-function's block.

G203 causes G1, G63 and G90 to become active.

The contour description can start at any point, e.g. the middle of a side's length.

The sides of the contour can be described using all the possibilities of the control to indicate linear and circular sides.

The pocket contour must be closed; if a gap is present the CNC will automatically close that gap with a linear contour side.

Absolute or incremental coordinates are allowed in the contour description. Programming is effected via G90 or G91.

The geometric function (G64) may be used. The same conditions will apply as for programming a geometric contour. The most important is that between G64 and G63 absolute coordinates must be used exclusively.

G203 START POCKET CONTOUR DESCRIPTION

Only axes in the main plane may be programmed. The program can start its finishing cut from the middle of a side's length; this is done by positioning the start point of the pocket cycle description at the middle of the side.

The program starts its finishing cut at the start point of the contour description. The finishing cycle will follow the same sequence as the contour description.

Only the following functions are permitted between G203 and G204, or between G205 and G206:

G1, G2, G3, G208 G63, G64 G90, G91

The G1, G2/G3 movements are limited to the main plane. Tool-axis and rotary axis coordinates are not permitted.

The G63/G64 and G90/G91 functions must all be programmed in separate blocks e.g.:

N10 G4

N20 X... Y... :This is acceptable.

N10 G64 X... Y... :This is not acceptable.

The associated pair of functions G203/G204 must be written in the same program or the same macro.

The first point of a contour description must be given in a G203-block.

Defined points may be used in a contour, but a point definition (G78) is not allowed. The points must have been defined prior to the G200-block of the pocket.

The bottom of the pocket must be parallel to the machining plane, i.e. XY-plane (G17), XZ-plane (G18) or YZ-plane (G19). A slanted or a curved bottom plane is not allowed.

The sides of the pocket must be perpendicular to the bottom plane.

Two parts of the same pocket may not intersect, or be tangent, to each other. However, two different pockets may intersect each other.

During finishing the programmer should make sure that the tool diameter is selected smaller than the distance to the narrowest part in pocket of the workpiece. The control system is unable to identify contours damaged during finishing.

The rules, which apply to G1 and G2/G3 movements, also apply to the same types of movement used within pocket cycles.

When the (G64) Geometry function is active, only absolute (G90) coordinates can be used and only one predefined (G78) point can be present in a program block.

Example Pocket with islands

N9990

N1 G54

N2 G17

N3 G195 X-10 Y-10 Z10 I320 J320 K-60

N4 G99 X0 Y0 Z0 I300 J300 K-40

N5 G200

N6 T2 M6 (predrilling start point, drill R10)

N7 G81 Y2 Z-20 F200 S3000 M3

N8 G22 N=9992

N9 T3 M6 (clearing out the pocket, mill R8)

N10 S2500 M3

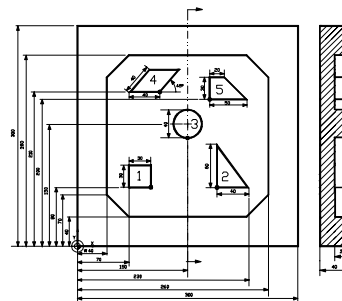
N11 G201 Y0.1 Z-20 B2 I50 K8 F200 F2=100 N1=9991 N2=9992

N12 G203 X40 Y40 Z0 N1=9993

:

N14 G204

N37 G202

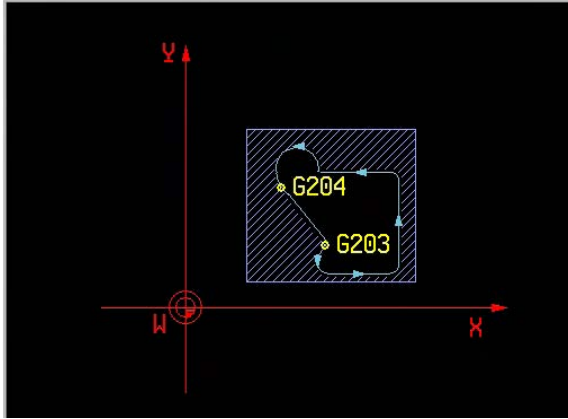


5.79 G204 End pocket contour description

End pocket contour description

Format

G204



Notes and Usage

G204 is only allowed to be programmed between G201 and G202.

Only the following functions are permitted between G203 and G204:

G1, G2, G3, G208, G63, G64, G90, G91

The G1, G2/G3 movements are limited to the main plane. Tool-axis and rotary axis coordinates are not permitted.

The G63/G64 and G90/G91 functions must all be programmed in separate blocks e.g.:

N10 G64

N20 X... Y... :This is acceptable.

N10 G64 X... Y... :This is not acceptable.

The associated pair of functions G203/G204 must be written in the same program or the same macro.

Example Pocket with islands

N9990

N1 G54

N2 G17

N3 G195 X-10 Y-10 Z10 I320 J320 K-60

N4 G99 X0 Y0 Z0 I300 J300 K-40

N5 G200

N6 T2 M6 (predrilling start point, drill R10)

N7 G81 Y2 Z-20 F200 S3000 M3

N8 G22 N=9992

N9 T3 M6 (clearing out the pocket, mill R8)

N10 S2500 M3

N11 G201 Y0.1 Z-20 B2 I50 K8 F200 F2=100 N1=9991 N2=9992

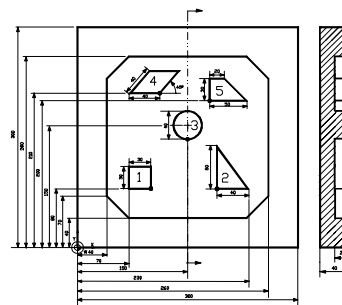
N12 G203 X40 Y40 Z0 N1=9993

N13 G208 X220 Y220 I30 (pocket contour)

N14 G204

:

N37 G202

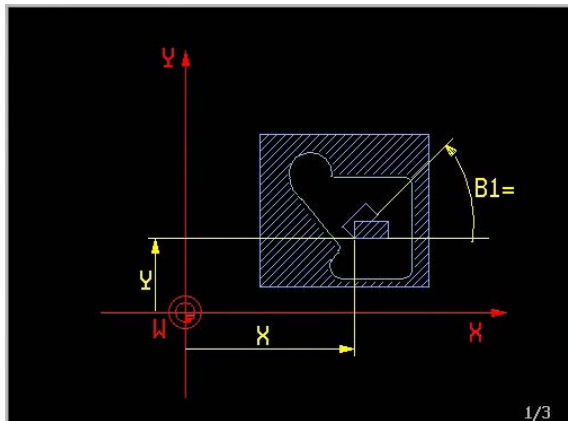


5.80 G205 Start island contour description

Start island contour description

Format

G205 X.. Y.. {N1=...}



G Start island contour description
 X Startpoint in X
 Y Startpoint in Y
 P Point definition number
 N1= Finishing macro number
 B1= Rotation angle of island contour
 B2= Start point polar angle
 L2= Start point polar length
 P1= Point definition number

- N1= Number of the macro for finishing the pocket contour. A finishing macro will not be generated if the word N1=.. is not programmed.
- B1= Rotation of the pocket contour around the point from the G203-block. Islands will not be rotated.

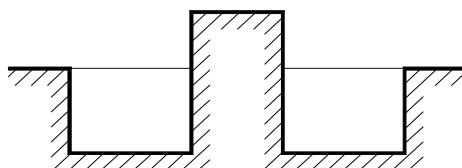
Description of the island contour

The contour of an island is described in the same way as the contour of a pocket. The description begins with G205 and the absolute starting position of the island.

The absolute position is described with either:

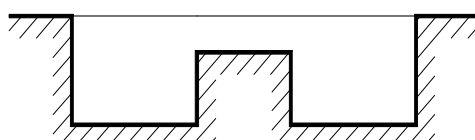
- Cartesian coordinates
- polar coordinates
- a defined point.

Programming of the tool axis is not allowed. The CNC assumes that the top surface of the island coincides with the top surface of the pocket.



If the island rises above the upper surface of the pocket, the B-word from the G201-block must be used to prevent a collision between cutting tool and workpiece during a movement from one starting point to the other.

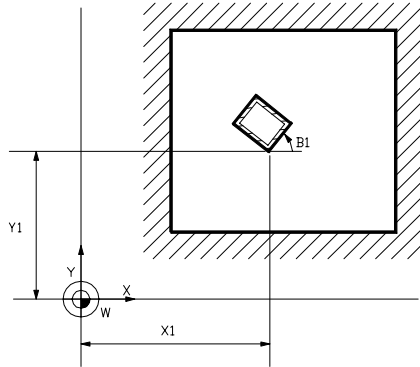
If the upper surface of the island lies below the upper surface of the pocket, the partprogrammer must see to it that, after the pocket has been cut, the island is brought to the correct level.



G205 causes G1, G63 and G90 to become active.
The contour description is terminated with G206.

For more information about island contour description see G203.

Rotation of an island around the starting point



The word B1=.. from the G205-block indicates that the island is rotated around its starting point (X1, Y1) through the programmed angle.

The programming will be:

```
G205  X (=X1) Y (=Y1) B1= (=B1)      N1=..
:                                     \
:                                     >..Island contour
:                                     /
G206
```

Notes and usage

Only the following functions are permitted between G205 and G206:

G1, G2, G3, G208 G63, G64 G90, G91

The G1, G2/G3 movements are limited to the main plane. Tool-axis and rotary axis coordinates are not permitted.

The G63/G64 and G90/G91 functions must all be programmed in separate blocks e.g.:

N10 G64

N20 X... Y... :This is acceptable.

N10 G64 X... Y... :This is not acceptable.

The associated pair of functions G205/G206 must be written in the same program or the same macro.

The contour of an island must be closed.

Two islands may not intersect with each other or be tangent.

Islands must be situated in the pocket and may not intersect with, or be tangent to, the pocket sides.

The sides of an island must be perpendicular to the bottom plane.

An island may not be enclosed by another island.

Example Pocket with islands**N9990**

N1 G54

N2 G17

N3 G195 X-10 Y-10 Z10 I320 J320 K-60

N4 G99 X0 Y0 Z0 I300 J300 K-40

N5 G200

N6 T2 M6 (predrilling start point, drill R10)

N7 G81 Y2 Z-20 F200 S3000 M3

N8 G22 N=9992

N9 T3 M6 (clearing out the pocket, mill R8)

N10 S2500 M3

N11 G201 Y0.1 Z-20 B2 I50 K8 F200 F2=100 N1=9991 N2=9992

N12 G203 X40 Y40 Z0 N1=9993

N13 G208 X220 Y220 I30 (pocket contour)

N14 G204

N15 G205 X100 Y80 N1=9994

N16 G208 X-30 Y30 J-1 (Island 1)

N17 G206

N18 G205 X190 Y80 N1=9995

N19 G91

N20 Y50 (Island 2)

N21 X40 Y-50

N22 G90

N23 G206

N24 G205 X150 Y130 N1=9996

N25 G2 I150 J150 (Island 3)

N26 G206

N27 G205 X110 Y210 N1=9997

N28 G208 X-40 Y40 J-1 B1=135 (Island 4)

N29 G206

N30 G205 X180 Y200 N1=9998

N31 G91

N32 Y30

N33 X20 (Island 5)

N34 X30 Y-30

N35 G90

N36 G206

N37 G202

N38 T4 M6 (clearing out the pocket, mill R8)

N39 F200 S2200 M3

N40 G22 N=9993

N41 G22 N=9994

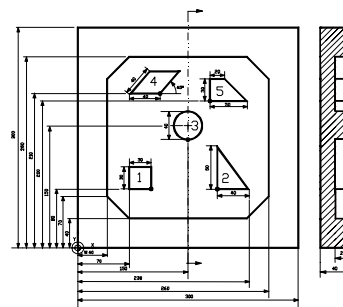
N42 G22 N=9995 (finishing)

N43 G22 N=9996

N44 G22 N=9997

N45 G22 N=9998

N46 M30

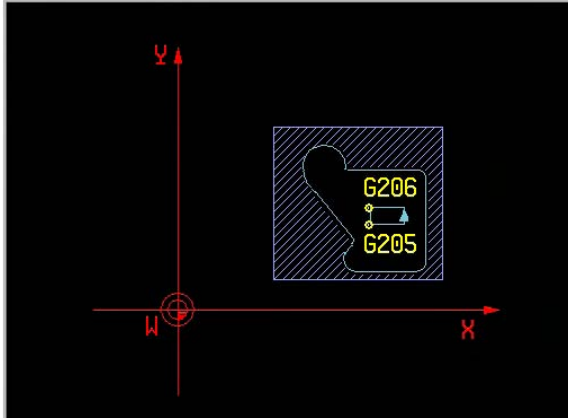


5.81 G206 End pocket contour description

End pocket contour description

Format

G206



Notes and Usage

Only the following functions are permitted between G205 and G206:
G1, G2, G3, G208, G63, G64, G90 and G91

The associated pair of functions G205/G206 must be written in the same program or the same macro.

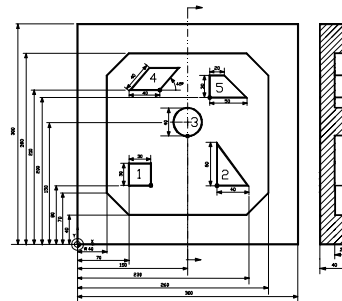
Example Pocket with islands

N9990

```

N1 G54
N2 G17
N3 G195 X-10 Y-10 Z10 I320 J320 K-60
N4 G99 X0 Y0 Z0 I300 J300 K-40
N5 G200
N6 T2 M6 (predrilling start point, drill R10)
N7 G81 Y2 Z-20 F200 S3000 M3
N8 G22 N=9992
N9 T3 M6 (clearing out the pocket, mill R8)
N10 S2500 M3
N11 G201 Y0.1 Z-20 B2 I50 K8 F200 F2=100 N1=9991 N2=9992
N12 G203 X40 Y40 Z0 N1=9993
N13 G208 X220 Y220 I30 (pocket contour)
N14 G204
N15 G205 X100 Y80 N1=9994
N16 G208 X-30 Y30 J-1 (Island 1)
N17 G206
N18 G205 X190 Y80 N1=9995
N19 G91
N20 Y50 (Island 2)
N21 X40 Y-50
N22 G90
N23 G206
N24 G205 X150 Y130 N1=9996
N25 G2 I150 J150 (Island 3)

```



N26 G206

N27 G205 X110 Y210 N1=9997

N28 G208 X-40 Y40 J-1 B1=135 (Island 4)

N29 G206

N30 G205 X180 Y200 N1=9998

N31 G91

N32 Y30

N33 X20 (Island 5)

N34 X30 Y-30

N35 G90

N36 G206

N37 G202

N38 T4 M6 (clearing out the pocket, mill R8)

N39 F200 S2200 M3

N40 G22 N=9993

N41 G22 N=9994

N42 G22 N=9995 | (finishing)

N43 G22 N=9996

N44 G22 N=9997

N45 G22 N=9998

N46 M30

5.82 G207 Call island contour macro

Programs the same island contour in another place

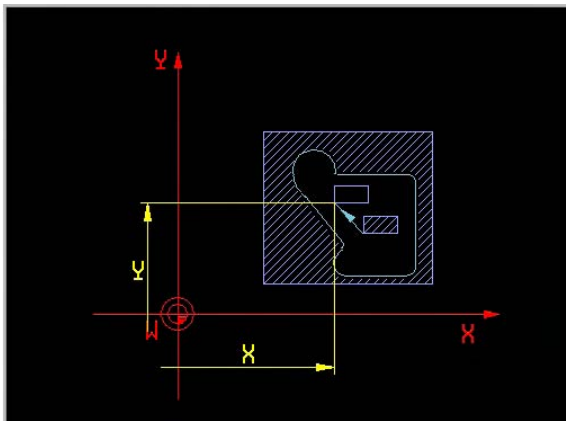
Three possibilities are available:

1. The same island occurs in another place within the same pocket contour.
2. The same island contour occurs in another pocket contour.
3. The same island contour occurs in another program.

By including the island contour in a macro the three possibilities can be treated in the same way.

Format

G207 N= X... Y... {Z...} {N1=...}



```
G    Call island contour macro
X    Shift along in X
Y    Shift along in Y
Z    Shift along in Z
N=   Macro with islandcontour
N1=  Finishing macro number
```

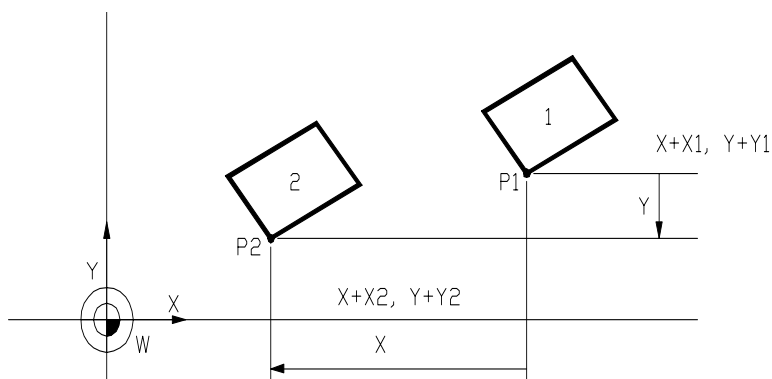
N=.. Macro number with island contour

X, Y The distance between the start point in the macro of the programmed contour and the desired start point

Notes and Usage

Geometric programmed contours (G64...G63) within a G207 macro (pocket cycle) may function incorrect because the macro-offset is only taken into account at the start point. Programmers using pocket cycle are advised to check the pocket macros on this phenomenon and if so to incorporate the macro concerned into the partprogram in order to overcome the problem.

Example



The macro of the island contour will then be:

N9xxx G205 X=X2 Y=Y2 N1=..

(X and Y are the start coordinates of the island contour in respect to the workpiece zero point.

```

N1          \
:           > Island contour
N..        /
N.. G206

```

Here, N9xxx is the identification of the macro.

The macro is called with the function G207.

N.. G201

N.. G207 N=9xxx N1=....

N.. G207 N=9xxx X=(X1-X2) Y=(Y1-Y2) N1=....

N.. G202

Explanation:

- 1 Island whose contour is programmed as a macro.
P1 Starting point of the contour description (G205-block).
- 2 Desired position of the island.
P2 Starting point of the shifted contour.
X.. Distance parallel to X-axis from P1 to P2.
Y.. Distance parallel to Y-axis from P1 to P2.
The distance carries a sign, as with incremental programming.

Note The best way is too start the island contour with the coordinates X0, Y0. (Zero point shift in G205). In the G207 block the start point can programmed, without calculating.

The same macro of the island contour is than:

N9xxx G205 X0 Y0 N1=..

```

N..          \
:           > Island contour with zero point shift
N..        /
N.. G206

```

N9xxx is the macro number.

The macro is called with the G207 function.

N.. G201

:

N.. G207 N=9xxx X=X2 Y=Y2

N.. G207 N=9xxx X=X1 Y=Y1

N.. G202

G63/G64 are not permitted in the macro.

Absolute and/or incremental programming is possible.

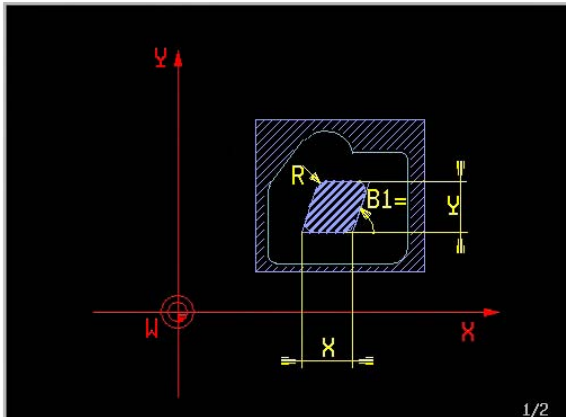
5.83 G208 Quadrangle contour description

To describe a regular rectangle as a pocket or island contour.

The G208 can be used to easily program a regular rectangle, more in particular a rectangle or parallelogram, as a contour.

Format

G208 X.. Y.. Z.. {I..} {J..} {R..} {B1=..}



G Quadrangle contour description
 X Length in X
 Y Length in Y
 Z Length in Z
 I Chamfer length
 J 1=climb, -1=conventional
 R Rounding radius
 B1= Angle quadrangle contour

N: Block number

X and Y These words state distances along the two main-plane axes. These distances are measured from the start point stated by the G203-function. The sign of each word states the direction in which a distance was measured. A '+' distance is measured in a positive axis direction and a '-' distance measured in a negative axis direction.

J The direction of the finishing movement will be determined by the J-address:

J1 Milling in counter clockwise direction (climb milling)

J-1 Milling in clockwise direction (conventional milling)

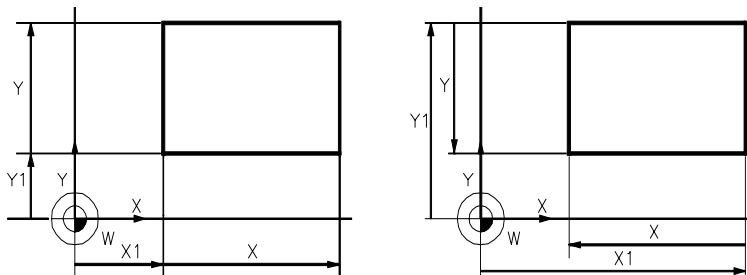
J is automatically 1, when no J-Address is programmed.

Notes and Usage

The bottom of the pocket must always be parallel to the main-plane.

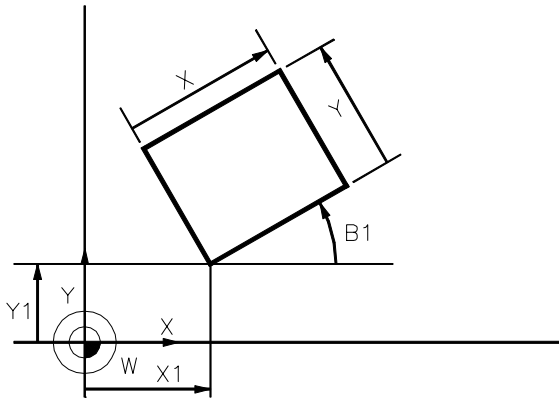
Examples

Example 1 Rectangle



If the sides of a rectangle are parallel to the axes, the X and Y coordinates are equally parallel to the axes.

If the sides are not parallel to the X and Y-axis, G208 is programmed as if the sides are parallel. The G203- block contains the rotation angle (B1=...).



Programming will be:

G203 X (=X1) Y (=Y1) Z (=Z1) B1= (=B1)

G208 X (=X) Y (=Y)

G204

Example 2 Parallelogram

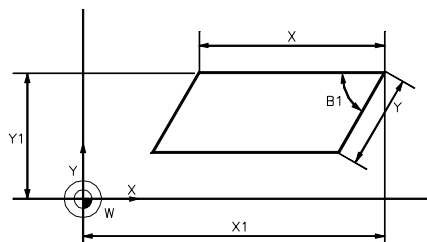
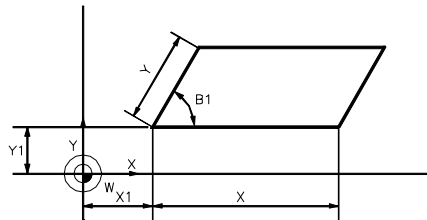
To program a parallelogram the lengths of both sides and the angle between them at the start point are stated.

Here:

B1=: The angle in degrees and decimal fractions of degrees ($0^\circ < B1 < 180^\circ$).

The angle carries no sign.

The default value for B1=.. is 90 degrees, i.e. a rectangle.



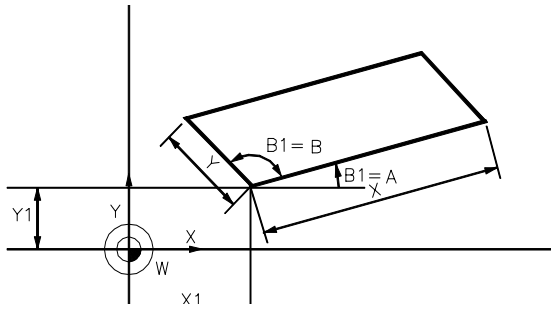
Programming will be:

G203 X (=X1) Y (=Y1) Z (=Z1)

G208 X (=X) Y (=Y) B1= (=B1)

G204

If one side is not parallel to the X-axis, programming will be the same. The G203-block contains the rotation angle.



Programming will be:

G203 X (=X1) Y (=Y1) Z (=Z1) B1= (=A)

G208 X (=X) Y (=Y) B1= (=B)

G204

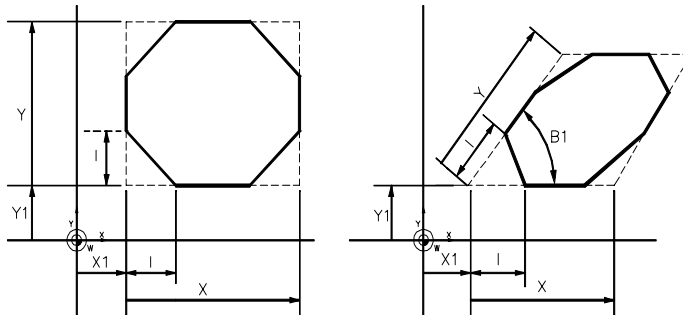
Example 3 One chamfer

For both the rectangle and the parallelogram a bevel or phase can be added.

I-word The width of the chamfer.

The I-word carries no sign.

The chamfer is symmetrically arranged around the corner.



Programming will be:

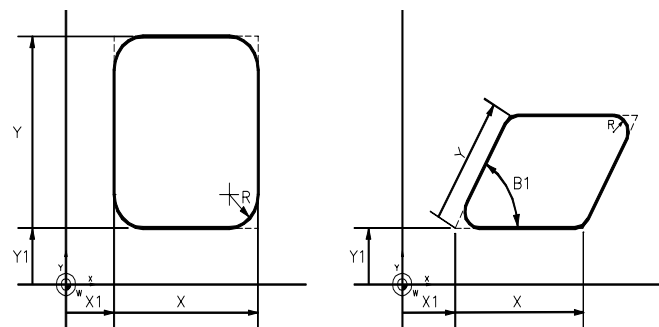
G203 X (=X1) Y (=Y1) Z (=Z1)

G208 X (=X) Y (=Y) B1= (=B1) I (=I)

G204

Example 4 Rounding

R-word The radius of the rounding. The radius does not have a '+/-' sign.



Programming will be:

G203 X (=X1) Y (=Y1) Z (=Z1)

G208 X (=X) Y (=Y) B1= (=B1) R (=R)

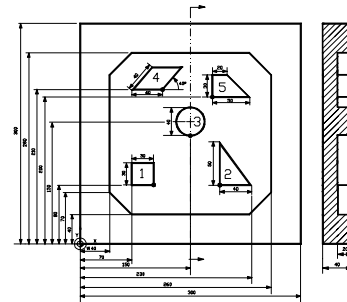
G204

Either the I-word or the R-word can be programmed: the two words cannot be used at the same time, if this is done, an 'invalidation contour description' message (P144) is generated.

Example 5 Pocket with islands

N9990

N1 G54
 N2 G17
 N3 G195 X-10 Y-10 Z10 I320 J320 K-60
 N4 G99 X0 Y0 Z0 I300 J300 K-40
 N5 G200
 N6 T2 M6 (predrilling start point, drill R10)
 N7 G81 Y2 Z-20 F200 S3000 M3
 N8 G22 N=9992
 N9 T3 M6 (clearing out the pocket, mill R8)
 N10 S2500 M3
 N11 G201 Y0.1 Z-20 B2 I50 K8 F200 F2=100 N1=9991 N2=9992
 N12 G203 X40 Y40 Z0 N1=9993



N13 G208 X220 Y220 I30 (pocket contour)

N14 G204

N15 G205 X100 Y80 N1=9994

N16 G208 X-30 Y30 J-1 (Island 1)

N17 G206

N18 G205 X190 Y80 N1=9995

N19 G91

N20 Y50 (Island 2)

N21 X40 Y-50

N22 G90

N23 G206

N24 G205 X150 Y130 N1=9996

N25 G2 I150 J150 (Island 3)

N26 G206

N27 G205 X110 Y210 N1=9997

N28 G208 X-40 Y40 J-1 B1=135 (Island 4)

N29 G206

N30 G205 X180 Y200 N1=9998

N31 G91

N32 Y30

N33 X20 (Island 5)

N34 X30 Y-30

N35 G90

N36 G206

N37 G202

N38 T4 M6 (clearing out the pocket, mill R8)

N39 F200 S2200 M3

N40 G22 N=9993

N41 G22 N=9994

N42 G22 N=9995 (finishing)

N43 G22 N=9996

N44 G22 N=9997

N45 G22 N=9998

N46 M30

5.84 G217/G218 Deactivate/Activate angular head

With G218 an angular head is activated. With this it is possible, also in a slanted plane (G7), to define correctly the dimensions and direction (plane) of an angular head with tool.

Format

G217
G218 {X} {Y} {Z} {A5=} {B5=} {C5=}

G	Activate angular head
X	Offset angular head
Y	Offset angular head
Z	Offset angular head
A5=	Rotation tool direction X-axis
B5=	Rotation tool direction Y-axis
C5=	Rotation tool direction Z-axis

X, Y, Z Defines the offset without tool in X, Y, Z-direction of the angular head [mm].
A5=, B5=, C5= Defines the rotation around the X, Y, Z-axis (space angle) of the tool direction (degr.). If no angle is programmed, a default value of A5= -90 [degr.] is taken. This corresponds with an angular head in negative Y-direction.

Notes and usage

Modality

G217 and G218 are mutual modal.

Deactivation

The function G218 is deactivated by G217.
G217 deactivates the allowances of G218. The normal tool length of the active tool is reactivated.
G217 and G218 refrain from all actions until the movements in the previous block are stopped with <INPOD>.

Data, used when activating the angular head.

- Dimensions of the angular head in X, Y, Z and tool direction in A5=, B5=, C5=.
- Tool length, radius and corner radius from the tool table. Also additional lengths and radii from the tool table are used.
- Depending on the IPLC, the angular heads have their own Q3= coding in the tool table.

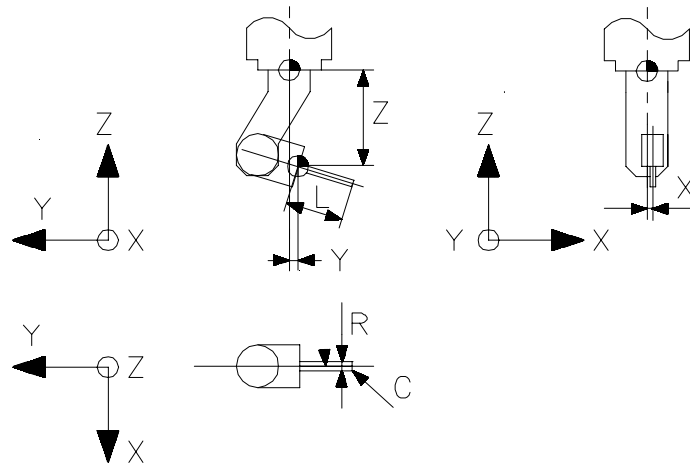
Note: For the measures of the angular head, it is assumed that the angle setting and the tool are fixed. The angle and the tool cannot be changed without measuring the complete system again.

Data of the angular head (Array).

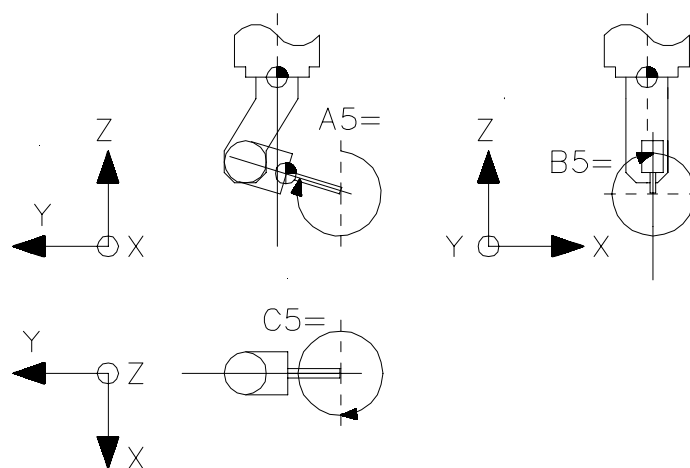
The data of the angular head are stored in arrays.
During measuring the angular head, the cycle writes the measured dimensions in an array.

Note: These cycles and the basic function G128 can also be used for a feed spindle.

Dimensional notations: Angular head reference point



Dimensional notations: Angular head direction = G7 plane



The offsets of the angular head are defined without a tool. The dimensions are defined in the positive direction, which means that the Z-offset is positive in any case and the X- and Y-offsets are depending on the situation (in this example positive).

The angles are defined as space angle. This means as a positive rotations around the positive normalised linear axes XYZ (as in G7). Herewith the rotation around the C-axis is executed first, then around B- and finally around the A-axis.

In this example applies: A5 = 290 or -70 [degr.]

B5 = 0 [degr.]

C5 = 0 [degr.]

Note: The angle C5= is measured from the positive X-axis. A default rotation between this positive X-axis and the M19 D0 position (and angle setting on the angular head) is set in a machine constant.

G7 Plane

When G218 is active, the plane must be set separately with G7. Herewith the G7 can be programmed with the same angles as defined for the angular head. In this case the rotary axes do not turn.

When required, the main plane (Xp, Yp) can be turned with G7 C6=.

Turning mode G36

In turning mode it is also possible to activate an angular head (anyhow in theory). In this case the tool radius R is also compensated with the angles of the angular head in the turning planes G17 Y1=1 Z1=2 and G18 Y1=1 Z1=-2.

Tool length allowance

When G218 is activated, the G39 "allowance programming" and the measuring cycle allowance L4= in the tool table are also compensated with the angles of the angular head.

Tool retract movement

When G218 is active, the G174 "tool retract movement" is executed in the direction of the angular head.

Note: When G174 is programmed with axis information, the real axis is moved as usual.

Start up of the CNC

G218 is immediately active after starting up the CNC. The function G218 is stored with parameters in the stand-by memory.

Display

The function G218 is not visible in the display

Kinematic model

The function is operative for all machine tool types.

Example: activating angular head

Program example	Description
N1 G218 X0.01 Y-25 Z150 A5=-60 B5= 0 C5= 0	Activating angular head
N2 G217	Deaktivieren Winkelkopf

5.85 G227/G228 Unbalance Monitor: ON/OFF

G227 Switch off Unbalance Monitor.

G228 Switch on Unbalance Monitor.

For the description, please refer to the chapter "Turning mode".

5.86 G240/G241 Contour check: OFF/ON

A contour can be checked in two ways with this function:

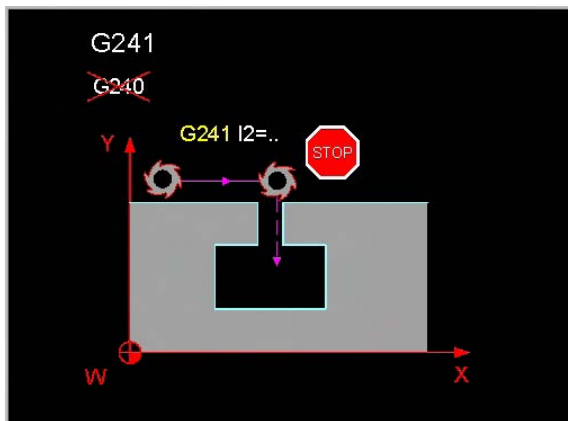
- 1 With the reverse check (I1=1 I2=0) is checked whether the compensated straight line (G0/G1) or circle or the programmed straight line (G0/G1) or circle are running in the same direction.
- 2 With the look ahead check (I1=0 I2=nnn) is checked whether the tool collides with the programmed contour.

These functions are only valid for programs with G41 and/or G42.

Format

G240

G241 {I1=} {I2=...}



```
G   Contour check: ON
I1= Reverse check: 0=off, 1=on
I2= Look ahead check: 0=off, >0=number
```

I1= Reverse check:

- 0 = no reverse check (compatible with previous versions).
- 1 = all movements with radius compensation are checked on "reverse".

I2= Defines whether this contour is checked with look ahead check:

- 0 = no check
- nnn = Number of blocks for look ahead check. When nnn > 0, the look ahead check is active.
Value lies between 0 and 400 (Default: nnn=5)

Note: In version V510, G241 without a parameter is the same as G241 I1=1.

In version V520, G241 without a parameter is the same as G241 I1=1 I2=5.

Note and usage

Refer also to G41/G42

Modality

G240 and G241 are mutual modal.

Deactivating

G241 will be deactivated with G240, M30, < CANCEL PROGRAM > or < CLEAR CONTROL >

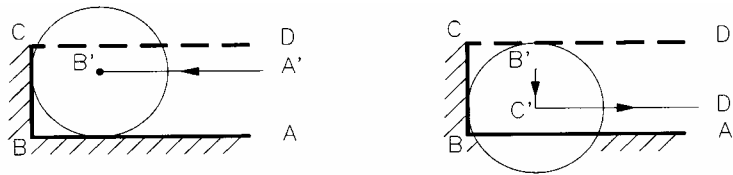
Programming errors

If an inversion of the direction is detected, an error message P412 is given.
<Corrected contour in wrong direction>

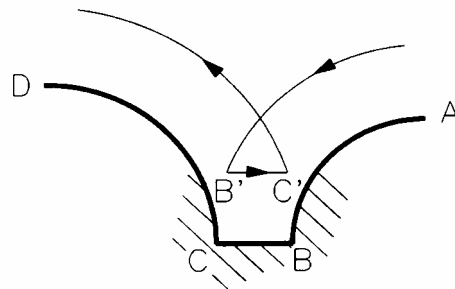
Direction inversion

When the radius of the tool is too big, an inversion of the direction can take place and the workpiece can be damaged. After activating G241 an error message is generated in this case.

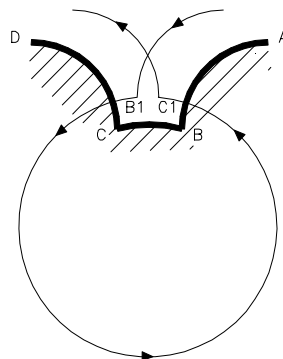
- a. The contour from AB to BC is programmed. With active radius compensation the tool retracted along CD. If BC is smaller than two times the tool radius, the tool collides with the workpiece during the movement from B' to C' and from C' to D'.



- b. A contour of the shape given in the illustration below is programmed. If the straight line is smaller than two times the tool radius, the tool collides with the workpiece during machining.



- c. A contour of the shape given in the illustration below is programmed. The tool moves to point B1, then from B1 to C1 and then parallel along CD. The movement from B1 to C1 takes place in the same direction as programmed on the circle BC. If the circular movement BC is too small, the tool will make almost a complete circle before it arrives at C1.



I2= Checking the contour with look ahead check:

Starting the look ahead check

G241 with parameter I2 > 0 sets a modal status. Herewith a look ahead check is started for every next block with G41 or G42.

stopping look ahead check

The look ahead check is stopped by:

- A block with G40, G240 or M30
- A block that switches the radius compensation off automatically (e.g. G79)
- A block with a programming error or a G function that is not allowed (error message)
- End of program or end of an internal read in macro (CAD-mode or BTR)
- Detected collision

Only when no collision is detected the machining of the contour is started.

Interrupt

The calculations for the G241 function can be interrupted.

After interrupting a checked contour, changing the program or tool measures and restarting, the changed contour is not checked again.

Programming errors

When the contour to be checked is faulty, the corresponding error message is already generated during contour checking, together with a P34 error message for the block number.

When during execution a collision is recognised, the error following error message is displayed:

P416 Collision N@@@@@ with N@@@@@

Example: P416 Collision N24 with N16

When milling block N16 block number N24 is damaged.

Performance

The calculation time for the algorithm of G241 I2= is proportional to the total number of the movement elements and to the number of movement elements (I2= parameter) that are checked against each other. The look ahead check of a contour of 100 blocks where 20 blocks must be checked against each other (I2=20) must be ready within 10 [sec.]

Display

The G241 function is shown in the modal G-group display.

During the calculations for the G241 function the "yellow clock" is displayed.

Graphics

When the G241 I2= function finds a collision during a graphical test run, the contour is drawn up to the colliding blocks. With the wire plot graphics the blocks are drawn with the block number and the erroneous block in yellow. The error message P416 is displayed in the last drawn block.

Note:

The display of block numbers in the wire plot graphics can also be turned on for "normal" cases. To activate this the softkey F4 <Block numbers> is added to the process <Execution>, menu <Options: Graphics>.

Manual block search

During manual block search the checking of the G241 function are carried out normally.

Example: Contour with radius compensation is checked with look ahead check

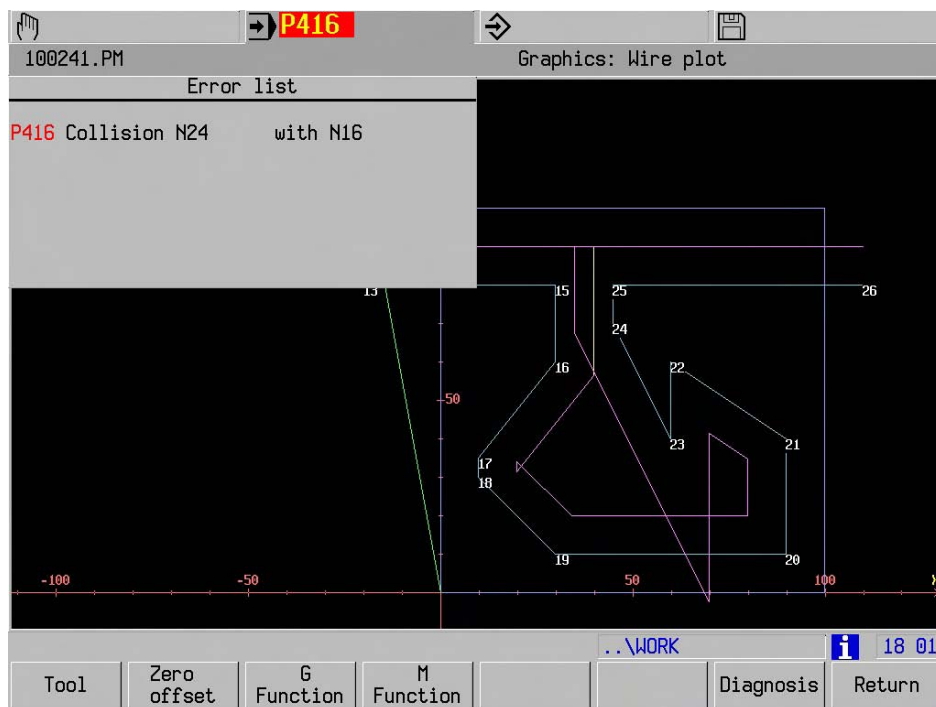
Programming example	Description
N100241	Program number
N1 G195 X-5 Y-5 Z5 I110 J110 K-30	Graphics window
N2 G199 X0 Y0 Z0 B1 C2	Material description
N3 G198 X0 Y0 Z0 D20	
N4 G1 X100	
N5 Y100	
N6 X0	
N7 Y0	
N8 G196	End of material description
N9 T20 M6 (Radius 10)	Tool definition radius 10 mm
N10 F1000 S1000 M3	Set feed and spindle speed
N11 G241 I1=0 I2=15	Switch on contour look ahead check (15 blocks)
N12 G0 X-20 Y110 Z-5	Starting position
N13 G43 X-20 Y80	
N14 G41	Switch on radius compensation
N15 G1 X30	Contour description
N16 Y60	
N17 X10 Y35	

N18 Y30	
N19 X30 Y10	
N20 X90	
N21 Y40	
N22 X60 Y60	
N23 Y40	
N24 X45 Y70	
N25 Y80	
N26 X110 Y80	Endposition Kontur
N27 G40	Radiuskorrektur ausschalten
N28 G240	Kontur vorausberechnen ausschalten
N29 M30	Programm Ende

The function G241 I2= builds internally a material contour of all the elementary movements, including the possible generated interconnection circles. After that is checked whether the tool wrap of every elementary movement is not colliding with the programmed number (I2=) of blocks of the look ahead check in the material contour.

The G241 I2= function is programmed modally and works only when the radius compensation is activated. The look ahead check is executed in every block with G41 or G42.

At the first found collision an error message is generated.



In this example 3 collisions are programmed.

The first collision is reported as error: P416 Collision N24 with N16. The other errors are not reported. These are collision N19 with N17 and collision N20 with N23.

In this case all collisions are avoided by reducing the cutter radius to 5 mm.

6. Specific G-Functions for macros

6.1 Overview G-Functions for macros:

Error message functions

- G300 Programming error messages
- G301 Error in a program or macro

Executable functions

- G302 Overwriting radius compensation parameters.
- G303 M19 with programmable direction
- G310 Store table on disk
- G311 Load table from disk

Query functions

- G318 Read pallet or job table data
- G319 Query actual technology data
- G320 Query actual G-data
- G321 Query tool data
- G322 Query machine constant memory
- G324 Query G-group
- G325 Query M-group
- G326 Query actual position
- G327 Query operation mode

Write functions

- G331 Write tool data

Calculation functions:

- G341 Calculation of G7-plane angles

Formatted write functions

- G350 Display window
- G351 Write to file

Array functions

6.2 Error message functions

6.2.1 G300 Programming error messages

Setting error messages during the execution of universal programs or macros.

Format

G300 [{D...}]{D1=...} =...

G Program error call
D P Error message number
D1= R Error message number

Notes and usage

D are general milling error messages (P), D1= are error messages (R) in turning mode (G36)

The error messages only cover the existing P and R-errors (refer to Machine Manual).

Example Setting an error message if a programmed angle is not allowed.

```

N9999 (Macro for calculation of table rotations)
N11 (input parameter: E4: phi)
N100
N110 G29 I1 E30 N=180 E30=(E4>360) Compare if E4 > 360 degrees. If so, jump to N180
N120 G29 I1 E30 N=210 E30=(E4<0) Compare if E4 < 0 degree. If so, jump to N210
N150 G29 I1 E30 N=290 E30=1 Jump to 290 (0 <= E4 <= 360 degrees)
N160
N170 (error message: phi>360)
N180 G300 D190 (programmed value > maximum value)
                                Error message: programmed value > maximum value
                                Program should be ended and a modified E4 be entered

N190
N200 (error message: phi<0)
N210 G300 D191 (programmed value < minimum value)
                                Error message: programmed value < minimum value
                                Program should be ended and a modified E4 be entered

220
N290 Normal program

```

6.2.2 G301 Error in program or macro that just has been read in.

Error in program or macro block that just has been read in.

Format

G301 (O... Wrong original block)

Notes and usage

When the controller retrieves a program block or macro block and discovers an error it activates G301
Function G301 can only be active in an error stopped program or macro.

This function cannot insert in MDI.

The error texts are O errors. (Refer to Machine Manual).

Example

The program is stored on hard disk.
Program is made with a MC84=0.

```
N9999 (Program)
N1 G17
N2 G57
N3 T1 M6
N4 F200 S1000 M3
..
N99 M30
```

Error stops program in RAM.
Zero point shift extension MC84 > 0 is active.

N9999 (ERR*)(Program ...)

```
N1 G17
N2 G301 (O138 G57)
N3 T1 M6
N4 F200 S1000 M3
..
N99 M30
```

G301 explains that the program is false. G57 must be
G54 I3

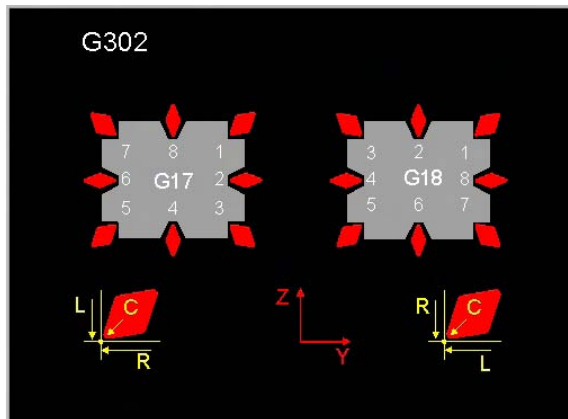
Note The false program can be activated. When passing the block G301 the controller stops and gives the following error text P33 (Modify block converted to connect). The block containing G301 must be changed before restarting.

6.3 Executable functions

6.3.1 G302 Overwriting radius compensation parameters

The G302 function overwrites the active tool parameters during execution. The tool parameters in the tool memory are not changed.

In this version, only the O parameter for tool orientation can be overwritten.



```
G  Ovrrule radius comp. parameters
0  Tool Orientation
```

For description refer to chapter "Turning mode".

6.3.2 G303 M19 with programmable direction

M19 with programmable direction (CW or CCW).

Format

G303 M19 D... I2=...

```
G  M19 with programmable direction
```

Notes and usage

Only M19 can be programmed.

Default for I2=3

Example

Stop spindle with M19.

N100 G303 M19 D75 I2=4

N100:Orientates spindle stop

Angle 75 degrees

CCW

6.3.3 G310 Store table on disk

Storing of user files such as parameter tables or tool data on hard disk.

The maximum number of lines in the user tables is limited by a maximum value allowed in machine constants. By saving to (G310) and reloading from (G311) hard disk of a part of or the complete table the number can be virtually increased.

For tool tables the management is improved. All data of the tools can be stored centrally (presetting device) and still be reached by the CNC.

Format

G310 N5= {I1=} {I2=}

```
G    Store table on disk
I1=  First record in table
I2=  Last record in table
N5=  File name of table
```

N5= Defines the filename and path with which the table must be stored on the internal hard disk or on an external PC.

The complete file name <path + name + type> must be entered between quotation marks ("").

I1= Defines the starting address of a file section.

Value lies between 0 and the end of the relevant user file.

If I2= is not programmed all data lines are stored from I1= onwards.

I2= Defines the end address of a file section.

Value lies between the starting address and the end of the relevant user file.

If I1= is not programmed, all data lines are stored up to and including I2=.

Path definition (N5=)

Work directory is D:\work\

The definition of the path on the internal hard disk is:

- N5= "param.pa" Data is written to the work directory as param.pa.
- N5= "test1\param.pa" Data is written to the subdirectory "test1" of the work directory as param.pa.
- N5= "\test2\param.pa" Starting with \ means that the data is written directly to the directory D:\test2 as param.pa.
- N5= "C:\test3\param.pa" Error message.

The definition of the path on a network:

- N5= "Z:\test4\param.pa" **SP-version:** The user file is stored via NFS (Network File System: See Technical Manual) in the directory Z:\test4 as param.pa.
DP-version: The user file is stored via the windows network in the directory Z:\test4\ as param.pa.
- N5= "\\server1\test5\param.pa" **SP-version:** Error message.
DP-version: Starting with \\ means that the user file is stored via the windows network in directory \\server1\test5 as param.pa

The total length of the path and name is:

SP-version: 80 Characters

DP-version: 120 Characters

A local path may be only 5 directories deep in both versions

Note: The path definition is the same for WinShape as for the **DP-version**. The work directory however, depends on the installation, normally it is <c:\winshape\>.

Notes and usage of G310 and G311

Table type:

The following file types are allowed.

PA	E-parameter	Depending on MC83 (Number of E-parameters).
PT	Points	Depending on MC82 (Number of point definitions).'
TM	Tools	Only tool data outside the tool magazine. Depending on MC27 (Number of tools) and MC28 (Number of tool places in the magazine).'
Other tables		Only for service purposes. See Technical Manual.

Execution

G310 and G311 refrain from all actions until <INPOD>. G310 stores the specified section of the table on the hard disk

G311 reads the specified section of the table and stores it in memory. In the remaining program execution the new stored data is used immediately.

When reading the tool memory (G311), MC 774 (Tool in (0,1=clear, 2=protect, 3=replace) is taken into consideration.

Allowed G-functions

G310 and G311 are not allowed with G41, G42, G64 and G141.

Operation and display

When G310 or G311 are executed, the sofkey operation concerning the file functions of the tables is possible. On the other hand the functions G310 and G311 are executed when the file functions are used.

During the execution of G310 or G311 a "yellow clock" is displayed.

Graphics, test run

In the operation modes graphics and test run the functions G310 and G311 are executed.

Manual block search

During manual block search the functions G310 and G311 are executed.

Interrupt

G310 and G311 can be interrupted by <Feed Hold> and <Feed Speed Hold>.

Example:

Programming example	Description
N9000 (Loading/storing data)	
N1 E2=50	nnn=50 enter value
N2 E(E2)=E2	Ennn =nnn
N3 E2=E2+1	increase nnn with 1
N4 G29 I-1 N=2 E0=(E2<=250)	When nnn is equal to or smaller than 250 jump back to N2
N10 G310 N5="datei1.pa" I1=50 I2=250	Storing E-parameters 50 up to 250 on the directory D:\work in the file datei1.pa
N20 G311 N5="\\Server\MillPlus\Param.pa"	Adding of E-parameters in SRAM via network from the file Param.pa on the external directory "\\Server\MillPlus"

6.3.4 G311 Load table from disk

Loading user files such as parameter table or tool data from hard disk.

Note: Please read the description of G310 (Store table on disk) also.

Format

G311 N5= {I1=} {I2=}

G Load table from disk
I1= First record in table
I2= Last record in table
N5= File name of table

N5= File name and path, with which the table is stored on the hard disk. The complete file name <path + name + type> must be entered between quotation marks ("").

I1= Defines the starting address of a file section.

Value lies between 0 and the end of the relevant user file.

If I2= is not programmed all data lines are read from I1= onwards.

I2= Defines the end address of a file section.

Value lies between the starting address and the end of the relevant user file.

If I1= is not programmed, all data lines are read up to I2=.

Path definition (N5=)

Work directory is D:\work\

The definition of the path on the internal hard disk is:

- N5= "param.pa" Data is read from the work directory as param.pa.
- N5= "test1\param.pa" Data is read from the subdirectory "test1" of the work directory as param.pa.
- N5= "\test2\param.pa" Starting with \ means that the data is read directly from the directory D:\test2 as param.pa.
- N5= "C:\test3\param.pa" Error message.

The definition of the path on a network:

- N5= "Z:\test4\param.pa" **SP**-version: The user file is read via NFS (Network File System: See Technical Manual) from the directory Z:\test4 as param.pa.
DP-version: The user file is read via the windows network from the directory Z:\test4 as param.pa.
- N5= "\\server1\test5\param.pa" **SP**-version: Error message.
DP-version: Starting with \\ means that the user file is read via the windows network from directory \\server1\test5 as param.pa

Example:

Programming example	Description
	Work directory is D:\WORK\
N10 G311 N5="test1\param.pa"	File from D:\WORK\TEST1\ is loaded
N20 G311 N5="\test2\param.pa"	File from D:\TEST2\ is loaded
N30 G311 N5="c:\test3\param.pa"	Error message
N40 G311 N5="z:\test4\param.pa"	SP : File from NFS-directory Z:\TEST4\ is loaded. DP and WinShape: File from windows network directory Z:\TEST4\ is loaded
N50 G311 N5="//server1\test5\param.pa"	SP : Error message. DP and WinShape: File from windows network directory \\SERVER1\TEST5\ is loaded

6.4 Query functions

6.4.1 G318 Read pallet or job table data

Query pallet data or job table data.

Format

G318 I1=.. I2=.. I3=.. E...

```
G  Read pallet or job table data
E  E-parameter
I1= 1=Pallet manag. 2=Job admin.
I2= Index number of table record
I3= Table address 1-5=PQSP1L1/SFDRx
```

Possible function:

I1=1 Pallet management
 I2=.. Index number in pallet table. (PO.PO)
 I3=1 Pallet number
 I3=2 Priority
 I3=3 Workpiece status
 (0= empty, 1=blank, 2=cutting, 3=ready, 4=reject)
 I3=4 Pallet type
 I3=5 Location type

I1=2 Job administration
 I2=.. Index number in job table. (JA.JA)
 I3=1 Order size
 I3=2 Finished products
 I3=3 Defect products
 I3=4 Blanks

Notes and usage

Reading of addresses without data

If the address not exist, the E-parameter contains the number -999999999.

Example Query job administration and storing the data in E-parameter 10.

N... G318 I1=2 I2=5 I3=2 E10 I1=1 I2=5 I3=2 query of the number of finished products.
 E10 contains the number of finished products.

6.4.2 G319 Query actual technology data

Query active F (Feed), S (Speed), S1 (Cutting speed/rotational speed) or T (Tool number).

Format

G319 I1=.. E... {I2=..}

```
G  Read actual technology data
E  E-parameter
I1= 1-7 (F,S,T,S1,F1,F3,F4)
I2= 0=programmed, 1=actual
```

Possible function:

I1=1 Feed (F)
 I1=2 Speed (S)
 I1=3 Tool number (T)

I1=4	Cutting speed/speed (S1=) (only turning)
I1=5	Constant cutting feed (F1= by G41/G42)
I1=6	In depth feed (Infeed F3=)
I1=7	In plane feed (F4=)
I2=0	Programmed value (default)
I2=1	Actual value.

Notes and usage

Reading of addresses without data

If the address not exist, the E-parameter contains the number -999999999.

Example query active feed and storing the data in E-parameter 10.

N... G319 I1=1 E10 I2=0

I1=1 query feed.

E10 then contains the value

6.4.3 G320 Query current G data

Query address value of current modal G function and save this value in the E parameter provided for this purpose.

Format

G320 I1=.. E...

G	Read actual G data
E	E-parameter
I1=	Selection number

Notes and usage

Defaults

All values are initialised when the machine is started. Most parameters are set on zero.

Reading active modal g-functions

G324 can be used to query whether a G function is active.

Particular information can always be queried with G320.

Result dimension

The unit of the result is mm or inches. Degrees for angles.

Selection number

	G-function	result	default
	I1=selection number	min—max.	
	G7 Tilting working plane		
1	Angle of rotation A-axis	-180--180°	0
2	Angle of rotation B-axis	-180--180°	0
3	Angle of rotation C-axis	-180--180°	0
	G8 Tilting tool orientation		
4	Angle of rotation A-axis	-180--180°	0
5	Angle of rotation B-axis	-180--180°	0
6	Angle of rotation C-axis	-180--180°	0

G9 Defining pole position point		
7	Pole coordinate X-axis	0
8	Pole coordinate Y-axis	0
9	Pole coordinate Z-axis	0
Result from G17, G18, G19, G180 and G182		
10	First main axis	1--6
11	Second main axis	1--6
12	Tool axis	1--3
		1=X, 2=Y, 3=Z, 4=A, 5=B, 6=C
G25 Feed- and speed override active		
13	Feed- and speed override active	0
G26 Feed- and speed override not active		
13	Feed- and speed override not active	1--3
		1=F=100%, 2=S=100%, F und S=100%
G27 Positioning functions		
14	Feed movement (I3=)	0
15	Rapid movement (I4=)	0
16	Positioning logic (I5=0)	0
17	Acceleration reduction (I6=)	100%
18	Contour tolerance (I7=0)	MC765
G28 Positioning functions		
14	Feed movement (I3=)	0--1
15	Rapid movement (I4=)	0--1
16	Positioning logic (I5=0)	0--1
17	Acceleration reduction (I6=)	5—100%
18	Contour tolerance (I7=0)	0—10.000µm or MC765
G39 Activate tool offset		
19	Tool length offset (L)	0
20	Tool radius offset (R)	0
G52 Palettes zero point shift		
21	Zero point shift in X-axis	0
22	Zero point shift in Y-axis -	0
23	Zero point shift in Z-axis -	0
24	Zero point shift in A-axis -	0
25	Zero point shift in B-axis -	0
26	Zero point shift in C-axis -	0
G54 Standard zero point shift		
27	Zero point shift in X-axis -	0
28	Zero point shift in Y-axis -	0
29	Zero point shift in Z-axis -	0
30	Zero point shift in A-axis -	0
31	Zero point shift in B-axis -	0
32	Zero point shift in C-axis -	0
33	Angle of rotation	0

	G92/G93	incremental or absolute zero point shift	
34	Zero point shift in X-axis -		0
35	Zero point shift in Y-axis -		0
36	Zero point shift in Z-axis -		0
37	Zero point shift in A-axis -		0
38	Zero point shift in B-axis -		0
39	Zero point shift in C-axis -		0
40	Angle of rotation		0
	United zero point shift (G52 + G54 + G92/G93)		
41	Zero point shift in X-axis -		0
42	Zero point shift in Y-axis		0
43	Zero point shift in Z-axis -		0
44	Zero point shift in A-axis		0
45	Zero point shift in B-axis -		0
46	Zero point shift in C-axis -		0
47	Angle of rotation		0
	G72	Mirror image and scaling not active	
48	Scaling factor plane (A4=)	1	
49	Scaling factor tool axis (A4=)	1	
50	Mirror image in X-axis	1	
51	Mirror image in Y-axis	1	
52	Mirror image in Z-axis	1	
53	Mirror image in A-axis	1	
54	Mirror image in B-axis	1	
55	Mirror image in C-axis	1	
	G73	Mirror image and scaling active	
48	Scaling (factor or %) plane (A4=)	1	
49	Scaling (factor or %) tool axis (A4=)	1	
	MC714	0= Machining plane (factor) 1= Machining plane (percent eel) 2= all linear axes (factor) 3= all linear axes (percent)	
50	Mirror image in X-axis	-1--1	
51	Mirror image in Y-axis	-1--1	
52	Mirror image in Z-axis	-1--1	
53	Mirror image in A-axis	-1--1	
54	Mirror image in B-axis	-1--1	
55	Mirror image in C-axis	-1--1	
	System axes number determinate by machine constants (MC103, MC105, etc.).		
56	First main axis	0--6 =not active, 1=X, 2=Y, 3=Z, 4=A, 5=B, 6=C	
57	Second main axis	0--6	
58	Tool axis	0--6	
59	First rotation axis	0--6	
60	Second rotation axis	0--6	
61	Third rotation axis	0--6	
	Information of actual tools		
	(Value is zero, when T0 is active or no value is given):		
62	Actual tool length	(L/L1=/L2= + L4= + G39 L)	
63	Actual tool radius	(R/R1=/R2= + R4= + G39 R)	
64	Actual tool corner radius	(C)	
65	Actual tool orientation	(O or G302 O)	

- Actual spindle position angle after tool head rotation (G7 or manual)
- 66 Projected actual spindle position angle on the XY-plane after automatic (G7) or manual tool head rotation.
- G106 and G108 Kinematics calculations
- 67 Total shift in X (Rotary axis position - compensation + Kin. compensation - MC3x14,
- without programmable offsets
G108-offsets (head and table)
IPLC-offsets
- 68 Total shift in Y
- 69 Total shift in Z
- 70 Value from I1= address from G108
0 = G106 active
1 = G108 active (in the head and possibly in the table)
- G153 und G154 work piece zero point tracking
- 71 Programmed status
0 = G153
1 = G154
- G125 and G126 Programmed tool lifting
- 72 Programmed status
0 = G125
1 = PLC (G126 I1=1)
2 = INT (G126 I2=1)
3 = PLC + INT (G126 I1=1 I2=1)
4 = ERR (G126 I3=1)
5 = PLC + ERR (G126 I1=1 I3=1)
6 = INT + ERR (G126 I1=1 I3=1)
7 = PLC + INT + ERR (G126 I1=1 I2=1 I3=1)
- 73 Programmed distance
- Kinematic position of the rotary axis
- 74 Returns the kinematic position of the A-rotary axis
- 75 Returns the kinematic position of the B-rotary axis
- 76 Returns the kinematic position of the C-rotary axis
- 0 = not present
- 10 = controlled axis in the tool head
- 11 = controlled axis 45° in the tool head
- 12 = manual axis in the tool head (MC501 = 10n)
- 13 = manual axis 45° in the tool head (MC501 = 10n)
- 14 = swivel axis in the tool head (MC501 = 20n)
- 15 = swivel axis 45° in the tool head (MC501 = 20n)
- 20 = controlled axis in the work piece table
- 21 = controlled axis 45° in the work piece table
- 22 = manual axis in the work piece table (MC501 = 10n)
- 22 = manual axis 45° in the work piece table (MC501 = 10n)
- 23 = swivel axis in the work piece table (MC501 = 20n)
- 23 = swivel axis 45° in the work piece table (MC501 = 20n)
- Software endswitch
- 77 returns the distance to the positive SW-endswitch in X
- 78 returns the distance to the positive SW-endswitch in Y
- 79 returns the distance to the positive SW-endswitch in Z
- 80 returns the distance to the negative SW-endswitch in X
- 81 returns the distance to the negative SW-endswitch in Y
- 82 returns the distance to the negative SW-endswitch in Z

	G106 and G108	Kinematic calculations
83	G108 Offset in the X-axis	
84	G108 Offset in the Y-axis	
85	G108 Offset in the Z-axis	
	G153 and G154	work piece zero point tracking
86	G154 Offset in the X-axis	
87	G154 Offset in the Y-axis	
88	G154 Offset in the Z-axis	
	G218 activate angular head:	
89	G218 Offset in the X-axis	
90	G218 Offset in the Y-axis	
91	G218 Offset in the Z-axis	
92	G218 Rotation (space angle) in the A-direction	
93	G218 Rotation (space angle) in the B-direction	
94	G218 Rotation (space angle) in the C-direction	

Example Query of Address of G-function (I1=) and store of the value in E-parameter 10.

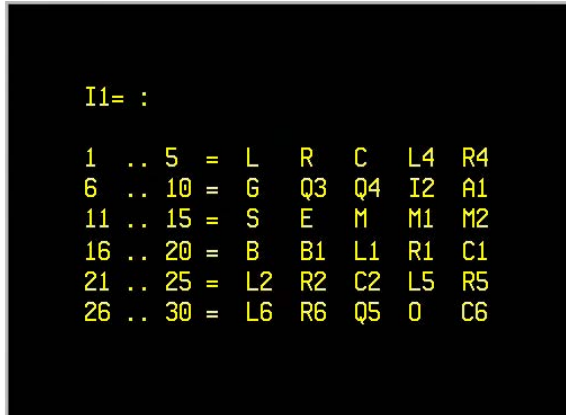
Programmbeispiel	Beschreibung
N11 G320 I1=10 E11	I1=10 Query first main axis E11 contains the result E11=1 X-axis is first main axis.
N12 G320 I1=11 E12	I1=11 Query second main axis E12 contains the result E12=2 Y-axis is second main axis.
N13 G320 I1=12 E13	I1=12 Query tool axis E13 contains the result E13=3 Z-axis is tool axis.

6.4.4 G321 Query tool data

Query tool table.

Format

G321 T.. I1=.. E...



I1= :					
1	..	5	=	L	R
6	..	10	=	G	Q3
11	..	15	=	S	E
16	..	20	=	B	B1
21	..	25	=	L2	R2
26	..	30	=	L6	R6

G Read tool data
 T Tool number
 E E-parameter
 I1= Tool address (1=L .. 30=C6)

Notes and usage

Tool number and position

The Tool number (T) must be known. The position (P) in the tool table cannot be queried.

Reading of the tool table values without data

If The E-Parameter contains the number -999999999, the address in the tool table is empty.

Classification

I1=1	L	Length
I1=2	R	Radius
I1=3	C	Corner radius
I1=4	L4=	Length oversize
I1=5	R4=	Radius oversize
I1=6	G	Graphics
I1=7	Q3=	Type
I1=8	Q4=	Number of cutting edges
I1=9	I2=	Cutting direction
I1=10	A1=	Approach angle
I1=11	S	Size
I1=12	E	Status
I1=13	M	Initial tool life
I1=14	M1=	Actual tool life
I1=15	M2=	Tool life monitoring
I1=16	B	Breakage tolerance
I1=17	B1=	Breakage monitoring
I1=18	L1=	First extra length
I1=19	R1=	First extra radius
I1=20	C1=	First extra corner radius
I1=21	L2=	Second extra length
I1=22	R2=	Second extra radius
I1=23	C2=	Second extra corner radius
I1=24	L5=	Wear tolerance length
I1=25	R5=	Wear tolerance radius
I1=26	L6=	Offset length
I1=27	R6=	Offset radius
I1=28	Q5=	Breakage monitoring cycle (0-9999)
I1=29	O	Tool orientation (only turning)

Example Program queries the tool table.

N30 G321 T10 I1=1 E1	G321 Read command T (tool number) I1= Information about the tool address E1 is E-parameter L (tool length) is set in E-parameter 1
N40 G321 T10 I1=2 E10	R (tool radius) is set in E-parameter 10
N50 G321 T10 I1=3 E20	C (corner radius) is set in E-parameter 20 (If C has no value, E20=-999999999 is set)
N60 G321 T10 I1=4 E2	L4 (length oversize) is set in E-parameter 2
N70 G321 T10 I1=5 E11	R4 (radius oversize) is set in E-parameter 11
N80 E3=E1+E2	The correct tool length (E3) is L+L4 (E1+E2)
N90 E12=E10+E11	The correct tool radius (E12) is R+R4 (E10+E11)

6.4.5 G322 Query machine constant memory

To read out a machine constant value and store it in the appropriate E-parameters.

Format

G322 E.. N1=...

G	Read machine constant memory
E	E-parameter
N1=	Machine constant number

Notes and usage

Reading out a machine constant without value

When invisible addresses are read from the machine constant table, the E-parameter remains unchanged.

Examples Universal program blocks, which can be used for both zero point, table types.

N50 G322 N1=84 E10	Machine constant 84 is set in E10
N60 G29 E1 N=90 E1=E10>0	Compare if MC84 > 0. If so, jump to N90
N70 G150 N1=57 X7=E1 Z7=E6	Store the zero point shift table ZO.ZO
N80 G29 E1 N=100 E1=1	Jump to N100
N90 G150 N1=54.3 X7=E1 Z7=E6	Store the zero point shift table ZE.ZE
N100 ..	

6.4.6 G324 Query G-group

Query current **modal** G-function and stores with this value in the E-Parameters preprogrammed for this purpose.

Format

G324 I1=.. E...

G Read G-group
E E-parameter
I1= G-group (1,2,etc.)

Notes and usage

Read out of group without data

If the group or the G-function not exists, the E-parameter is unchanged.

Group classification

I1=	G-function
1	G0, G1, G2, G3, G6, G9
2	G17, G18, G19
3	G40, G41, G42, G43, G44, G141
4	G53, G54, G54_I, G55, G56, G57, G58, G59
5	G64, G63
6	off, G81, G83, G84, G85, G86, G87, G88, G89, G98
7	G70, G71
8	G90, G91
10	G94, G95
11	G96, G97 (only turning)
12	G36, G37 (only turning)
13	G72, G73
14	G66, G67
15	off, G39
16	G51, G52
17	G196, G199
19	G27, G28
20	G25, G26, G26_S, G26_F_S
21	off, G9
22	G202, G201
24	G180, G182, G180_XZC
27	off, G7
28	off, G8
29	G106, G108

Result

In general is the result equal to the value of the modal G-function.

For example: G324 I1=3 gives, when G40 is active, as result the value 40.

Exceptions are:

Off gives value 0.

G26_S, G26_F_S gives 26.

G54_I gives 54.nn, where nn is the index.

G180_XYZ gives 180.

Example

selection of the G-function (I1=2) and storage of the value in E-parameter 10.

N... G324 I1=2 E10

I2=2: query group 2 G-function

E10 holds the result

E10 =17

G17 is active.

6.4.7 G325 Query M group

Query current modal M-function and store this value in the E-Parameter pre-programmed for this purpose.

Format

G325 I1=.. E...

G	Read M-group
E	E-parameter
I1=	M-group (1,2,etc.)

Notes and usage

Read out of group without data

If the group or the M-function does not exist, the E-parameter is unchanged.

Meaning M-functions

Some of these M-functions are basis M-functions and are described in the paragraph "M-functions" of chapter "Technological instructions". The other are machine dependent M-functions. Please refer to the machine builder handbook for a description.

Combined M-functions (M13 and M14)

M13 and M14 are combined M-functions. (M13=M3 + M8). These functions are determinate by two blocks.

N... G325 I1=1 E10.

N... G325 I1=3 E11

When E10=3 and E11=8, than M13 is active.

Group classification

Group	
I1=	M-function
1	off, M5, M3, M4, M19
2	off, M40, M41, M42, M43, M44
3	M9, M7, M8
4	off, M17, M18, M19
5	off, M10, M11
6	off, M22, M23
7	off, M32, M33
8	off, M55
9	off, M51, M52
10	off, M53, M54
11	off, M56, M57, M58
12	off, M72, M73
13	off, M1=..

Result

In general is the result equal to the value of the modal M-function.

For example: G324 I1=2 gives, when M40 is active, as result the value 40.

Exceptions are: Off gives value 0.

Example: selection of the M-function (I1=1) and storage of its value in E-parameter 10.

N... G325 I1=1 E10

I2=1: query group 1 M-function

E10 holds the result

E10 =5 M5 is active.

6.4.8 G326 Query actual position

To read out the actual axes-positions values and store it in the appropriate E-parameters.

Format

G326 {X7=..} {Y7=..} {Z7=..} {A7=..} {B7=..} {C7=..} {D7=..} {I1=..} {I2=..}

G Read actual position
 X7= E-parameter for X-position
 Y7= E-parameter for Y-position
 Z7= E-parameter for Z-position
 A7= E-parameter for A-position
 B7= E-parameter for B-position
 C7= E-parameter for C-position
 I1= 0=Workpiece 1=Machine 2=RPF
 I2= 0=programmed, 1=actual
 D7= E-parameter for S-position

I1=	0	Position to work piece zero point (Default)
	1	Position to machine zero point
	2	Position to reference point
	3	Total zero point shift (without IPLC shift).
I2=	0	Programmed value (default)
	1	Current value

Notes and usage

Reading out of not existent axes

When an axis not exist the contents of the E-parameter is filled with -999999999.

Reading out by graphical simulation

By graphical simulation only the X, Y and Z can be read out. The E-parameters for the rotating axes stays zero.

Reading out of spindle position (D7=):

When I1=0 is, is the result, the programmed spindle position of M19 or the programmed spindle position in G700.

Examples

Example 1: Read out actual axes-position von X, Y and Z and store the values in E-parameters 20, 21 and 22.

N... G326 X7=20 Y7=21 Z7=22 E20 contains the actual X-axis-position.

Example 2: Program continuation after a universal pocket cycle.

N30 G202	End pocket cycle
N40 G326 X7=20 Y7=21	Unknown actual End-position von X and Y
N50 G29 E1 N=90 E1=E20>100	Actual X-position >100, then jump to N90
N60 G29 E1 N=90 E1=E20<-100	Actual X-position <-100, then jump to N90
N70 G0 X-110	G0 movement to X-110, if the actual X-position is situated between 100 and -100. On this manner for example an obstacle can be rounded.
N80 G0 Y 100	Further turn aside movement

6.4.9 G327 Query operation mode

To scan the current operating mode and store this value in the E parameter provided.

Format

G327 I1=.. E...

G Read operation mode
E E-parameter
I1= Active mode (1-6)

Notes and usage

Arrangement of group

Group

I1=	Operating mode	
1	EASYoperate	0 = not active, 1=active
2	Single record	0 = not active, 1=active
3	Graphic	0 = not active, 1=active
4	Test run	0 = not active, 1=active
5	Search	0 = not active, 1=active
6	Demo	0 = not active, 1=active

Example Fetch operating mode (I1=1) and store the value in E parameter 10.

N... G327 I1=1 E10

I1=1: Check whether EASYoperate is active.

E10 contains the result: 0= not active, 1=active.

6.5 Write functions

6.5.1 G331 Write tool data

Write from values in the tool table.

Format

G331 T.. I1=.. E...

I1= :

```

1  .. 5  = L  R  C  L4 R4
6  .. 10 = G  Q3 Q4 I2 A1
11 .. 15 = S  E  M  M1 M2
16 .. 20 = B  B1 L1 R1 C1
21 .. 25 = L2 R2 C2 L5 R5
26 .. 30 = L6 R6 Q5 0  C6

```

```

G  Write tool data
T  Tool number
E  E-parameter
I1= Tool address (1=L .. 30=C6)

```

Notes and usage

Tool number and position

The tool number (T) must be known. The position (P) in the tool table cannot be changed.

Writing in the tool table without data

If the E-parameter contains the value -999999999, the address in the tool table becomes empty.

New information activating

The changed tool information must be activated again following the writing. (T.. M67)

Classification

I1=1	L	Length
I1=2	R	Radius
I1=3	C	Corner radius
I1=4	L4=	Length oversize
I1=5	R4=	Radius oversize
I1=6	G	Graphics
I1=7	Q3=	Type
I1=8	Q4=	Number of cutting edges
I1=9	I2=	Cutting direction
I1=10	A1=	Approach angle
I1=11	S	Size
I1=12	E	Status
I1=13	M	Initial tool life
I1=14	M1=	Actual tool life
I1=15	M2=	Tool life monitoring
I1=16	B	Breakage tolerance
I1=17	B1=	Breakage monitoring
I1=18	L1=	First extra length
I1=19	R1=	First extra radius
I1=20	C1=	First extra corner radius
I1=21	L2=	Second extra length
I1=22	R2=	Second extra radius

I1=23	C2=	Second extra corner radius
I1=24	L5=	Wear tolerance length
I1=25	R5=	Wear tolerance radius
I1=26	L6=	Offset length
I1=27	R6=	Offset radius
I1=28	Q5=	Breakage monitoring cycle (0-9999)
I1=29	O	Tool orientation (only turning)

The tool commentary cannot be changed.

Example

N10 E5=100 (Tool length)	L (tool length) is set in E-parameter 5
N11 E6=10 (Tool radius)	R (tool radius) is set in E-parameter 6
N12 E7=-999999999 (Tool corner radius)	C (tool corner radius) will be stored in E-parameter 7 (If C has no value, E7= must be set to -999999999)
N13 E8=0 (Length oversize)	L4 (length offset) is set in E-parameter 8
N14 E9=0 (Radius oversize)	R4 (radius offset) is set in E-Parameter 9
N..	
N20 G331 T10 I1=1 E5	L (tool length) writing of E-parameter 5 in the tool table
N21 G331 T10 I1=2 E6	R (tool radius) writing of E-parameter 6 in the tool table
N22 G331 T10 I1=3 E7	C (tool corner radius) writing of E-parameter 7 in the tool table
N23 G331 T10 I1=4 E8	L4 (length offset) writing of E-parameter 8 in the tool table
N24 G331 T10 I1=5 E9	R4 (radius offset) writing of E-parameter 9 in the tool table
N30 T10 M67	
The tool must be activated once more with the changed information.	
N..	
N40 E8=0.3 (Length oversize)	L4 (length offset) E-parameter 8 is set to 0.3
N41 G331 T10 I1=4 E8	L4 (length offset) writing of E-parameter 8 in the tool table
N50 T10 M67	
Tool must be activated once more with the changed information.	

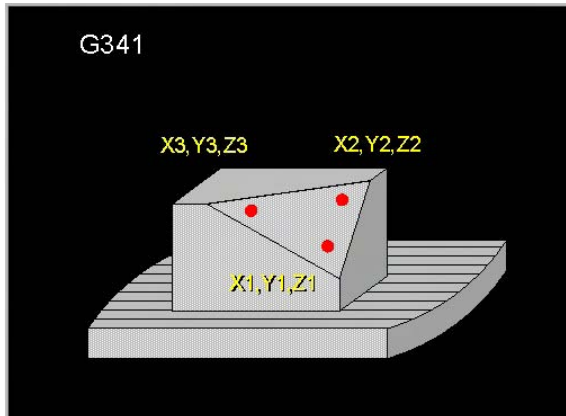
6.6 Calculation functions

6.6.1 G341 Calculation of G7-plane angles

G341 is used to calculate the solid angles A5=, B5= and C5= from 3 defined points. These angles are used in G7 to set up the plane.

Format

G321 {X1=.. Y1=.. Z1=.. X2=.. Y2=.. Z2=.. X3=.. Y3=.. Z3=..} O1=.. O2=.. O3=..



G	Calculation of G7 plane angles
X1=	E-parameter number of plane point
Y1=	E-parameter number of plane point
Z1=	E-parameter number of plane point
X2=	E-parameter number of plane point
Y2=	E-parameter number of plane point
Z2=	E-parameter number of plane point
X3=	E-parameter number of plane point
Y3=	E-parameter number of plane point
Z3=	E-parameter number of plane point
O1=	E-parameter number plane angle A5
O2=	E-parameter number plane angle B5
O3=	E-parameter number plane angle C5

Notes and usage

X1= to Z3= are E parameter numbers with axis position values of 3 points that define the machining plane [mm or inches]. If one of these addresses X1= to Z3= is programmed, all the addresses must be programmed. The 3 points do not have to be identical, nor do they need to be in a line. If the E parameters are not entered, G341 calculates A5=, B5= and C5= from the rotated plane that is set.

O1= to O3= are the numbers of the E parameters where the calculated solid angles A5=, B5= and C5= are stored [in degrees]. O1=, O2= and O3= must be programmed.

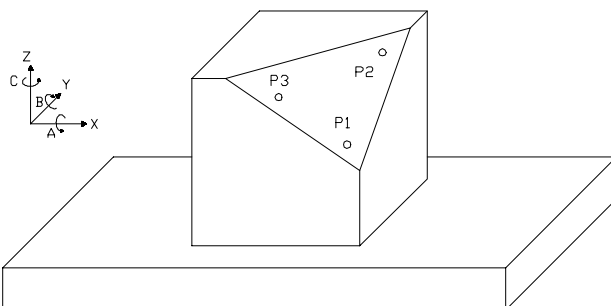
If G7 or G8 is active the input values must be defined in the active co-ordinate system.

G341 is not allowed if G19 is active.

Note

If the G341 inputs are determined in G7, G8, G17, or G18, the calculation by G341 must be carried out in the same mode.

Example: Flattening an oblique face.



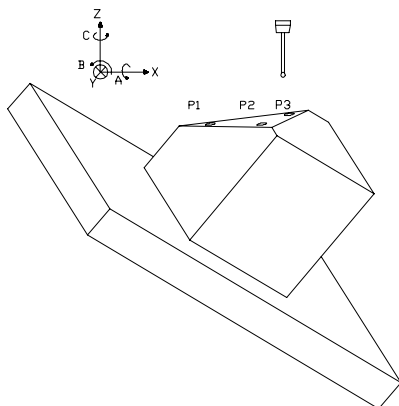
Therefore the oblique face must be defined by 3 points: (P1 (X,Y,Z), P2 (X,Y,Z) and P3 (X,Y,Z)). Because the face is too oblique to get accurate measure points, first the workpiece is turned until the oblique face has approximately been flattened (the round axes have been jogged and are not equal to zero anymore).

Next, the 3 points are determined with a measure probe and are saved in E-parameters E10 up to and including E18:

P1 (X, Y, Z) = E10, E11 and E12

P2 (X, Y, Z) = E13, E14 and E15

P3 (X, Y, Z) = E16, E17 and E18

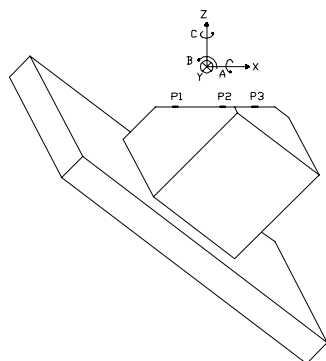


Next, G341 determines the round axes positions, which can be used by G7 to flatten the oblique face. The round axes positions are written in E-parameters E20, E21 and E22.

G341 X1=10 Y1=11Z2=17 Z3=18 O1=20 O2=21 O3=22

Finally the oblique face is flattened by G7:

G7 A5=E20 B5=E21 C5=E22



6.7 Formatted write functions

6.7.1 Introduction formatted write functions:

The formatted write function, can be used for:

- to write to the screen
- to write to the file on the hard disk

Configuration file to define a file or window (display/input).

Configuration files are required to describe how and where to write.

These configuration files are saved on the hard disk:

D:\STARTUP\CYCLES\FORMnnnn.CFG.

nnnn is the file number from 1 to 8999.

Configuration files are activated when the system is started.

End users can define files themselves.

The file size is unlimited.

Description of configuration file:

:Commentary starts with a ':'

;

;Sections:

Only for one window:

;[window] defines present window
;number= windowId where windowId = 1...4 See G350

;[file] defines file (only for G351')
;name = file name where filename is 8.3 ASCII characters
; The directory is always D:\STARTUP\

;[string] defines position and content of the block
;line = line number where line number = [1|...|n] basic setting = 1
;position= position number where position number = [1|...|n] basic setting = 1
;gb = "string" where block is <n> ASCII characters
;d = "string"

; Texts are defined for various languages
; Code gb=, d=, f=

; dependency condition (IF)

conditionparam= E-Parameter number [1|...|MC83] (e.g. 240)

conditionvalue = Values (e.g. 3)

When the 'conditionparam' (E240) has a 'conditionvalue' of 3, this instruction is executed. In this case the text "string" is written in the window or file.

;

;[value] defines position, format and E parameter of the value

;line = line number

;position= position number

;eparam= E parameter where E parameter number = [1|...|MC83]

;form = Determines the input format (default 6.3). 6.3 means: 6 figures before the decimal point and 3 after.

When the address dimension [mm], [degr], [mm/min] or [diam] is, the number of digits behind the decimal point depends of MC705 and MC707.

MC705 (Decimal digits behind the decimal point) is 3 or 4. The number or digits before and after the decimal point will be adapted.

MC707 (Inch/Metric). is 70 (metric) or 71 (Inch). When MC707=71 the number of digits behind the decimal point will be increased by one and the number of digits before the decimal point will be decreased by one.

Overview:	Metric		Inch	
MC707	71	71	70	70
MC705	3	4	3	4
Dimensions				
[mm] Linear axis	6.3	5.4	5.4	4.5
[degr] Rotation axis	6.3	5.4	6.3	5.4
[mm/min] Feed	6.3	6.3	5.4	5.4
[diam] Diameter programming in mm				
	6.3	5.4	5.4	4.5

;dimension= Only [mm], [degr], [mm/min] and [diam] are allowed. Addresses with these dimensions are depending of MC705 and MC707.

[mm] mm for linear axis
 [degr] Degree for rotation axis
 [mm/min] mm pro minute for feed
 [diam] Diameter programming in mm
 Default: no dimension

;sign = yesNo where yesNo = y = space for sign
 ; n = no space for sign

; Dependency condition (IF)

conditioneparam= E-Parameter number [1|...|MC83] (e.g. 240)

conditionvalue = Values (e.g. 3)

When the 'conditioneparam' (E240) has a 'conditionvalue' of 3, this instruction is executed. In this case the text "string" is written in the window or file.

;

Only for input window:

;[input] defines position, format and E parameter before an input field

; only for G350 and windowId = 2
 ; only one [input] section is allowed

;line = line number

;position= position number

;eparam= E parameter number where E parameter number = [1|...|MC83]

;form = digitDecimal where digitDecimal = <digits>.<decimals>

;sign = yesNo where yesNo = y =space for sign
 ; n = no space for sign

6.7.2 G350 Writing to a window

Specific lines and values can be written to a window using E parameters and a configuration file. In addition, a particular input can be expected. For unbalance detection, the result can be displayed to the operator in this way.

Format

G350 N1=.. {l1=...} {l2=...}

```
G      Write to window
N1=    Configuration file number
I1=    Window {0=closed, 1=open}
I2=    Window {0=no interv, 1=interv}
```

N1= Defines the configuration file 'D:\STARTUP\CYCLES\FORMnnnn.CFG> that is used for the format, lines and E parameters that are written. File number between 1 and 8999.

0 = window not visible. Setting on switch-on:
1 = window visible.

12= 0 = Program do not stop.
 1 = Program stops like "intervention" and waits for <Start>

Notes and application

G350 can be used to make a previously defined window visible. The texts in the window are fixed, and the values are continuously updated according to the defined E parameters.

When I2=1 is programmed, the program waits until <Start> is pressed. Only one entry window can be active at any one time.

4 windows are defined:

Number	Window type	Mode	Position	Size
1	Display	Manual Automatic	Right side of screen Top 'Dashboard'	15 lines, 37 characters
2	Input	Manual Automatic	Right side of screen Top 'Dashboard'	5 lines, 37 characters
3	Graphics	Manual Automatic	Left side of screen Top 'Dashboard'	
4	Display	Manual Automatic	Left side of screen Top 'Dashboard'	15 lines, 37 characters

The window also appears in graphics, but not during block search.

The window becomes invisible following M30 and <Cancel program>.

6.7.2.1 Writing to a window

N1 E11=45 Hole number

N2 E12=6 Number

File D:\STARTUP\CYCLES\FORM3501.CFG is used

Drilling pattern	
Maximum number of holes	45
hole number	6

Display window configuration file

;FORM3501.CFG

[Window]
 number = 1 ;Uses window number 1 of the available windows.

[string]
 line = 2
 gb = "drilling pattern"

[string]
 line = 4
 position = 1
 gb = "Maximum number of holes"

[value]
 line = 4
 position = 27 ;Print value in field at position 8 and onwards
 eparam = 11 ;E parameter E300 is given the value
 form = 3.0 ;format 3 figures and 0 decimals
 sign = n ;No space reserved for sign

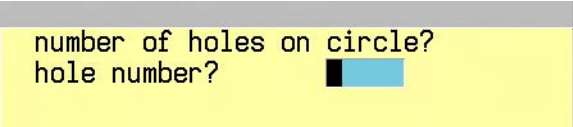
[string]
 line = 5
 position = 1
 gb = "Actual hole number"

[value]
 line = 5
 position = 27 ; Print value at position 27 and onwards
 eparam = 12
 form = 3.0
 sign = n

6.7.2.2 Writing to a window and asking for information

N10.. G350 N1=3502 I1=1 Write to window

File D:\STARTUP\CYCLES\FORM3502.CFG is used



```

number of holes on circle?
hole number?
  
```

Display window configuration file

;FORM3502.CFG

[window]
 number = 2 ; Uses window number 2 of the available windows.

[string]
 line = 1
 position = 1
 gb = "number of holes on circle"

[string]
 line = 2
 position = 1
 gb = "number of holes"

[input]
 eparam = 10 ;E parameter E10 contains an input value received the operator input
 form = 3.0 ;format 3 figures and 0 decimals
 sign = n ; No space reserved for sign

6.7.3 G351 Writing to a file

Specific lines and values can be written to a text file in D:\Startup\ using E parameters and a configuration file. This can be used to create the calibration curves for unbalance detection.

Format

G351 N1=.. {I1=...}

```
G    Write to file
N1=  Configuration file number
I1=  0=Add, 1=Overwrite
```

- N1= Defines the configuration file <'Directory'\FORMnnnn.CFG> that is used for the format, lines and E parameters that are written. File number between 1 and 8999.
The directory can be any 'Cycle Design' directory.
The configuration file is the same as for writing to a window, but 'section' [window] and [input] are ignored.
- I1= States whether the data is to be inserted at the end of an existing file or whether a file that may exist is to be overwritten. Basic setting <0> for insertion.

Notes and application

G351 writes the lines and values of the configuration file and E parameters to the hard disk.
A maximum of 50 lines of 255 characters can be written at the same time.
The file is not written during graphics or block search.

Example Listing measurement data and writing to a file.
The radius of a pocket is measured in the program

The following data available in the E parameters must be listed:

N10 (measurement programmed in blocks N12 to N16)
N11 (in this case as example of just the results from e.g. measurement cycle G145)
N12 E50=34.1 (setpoint) (entered)
N13 E51=34.05 (lower tolerance) (entered)
N14 E52=34.15 (upper tolerance) (entered)
N15 E53=34.108 (actual value) (measured)
N16 E54=0.008 (difference) (calculated)

N20 G351 N1=0002 I1=0 (write file)
File D:\STARTUP\CYCLES\FORM0002.CFG is used.
I1=0 is insert

The file messdat.txt is:

```
Radius
Setpoint =      34.1
Lower tolerance =34.5
Upper tolerance =34.5
Actual value =   34.108
Difference =      0.008
*****
```

Configuration file for listing measurement data FORM0002.CFG

```

*****
;
; CFG file for writing measurement data
*****
;

;---- Name of file to be written to startup\ -----
[file]
name          = Messdat.txt

;---- Type of measurement -----
[string]
line          = 1
position      = 1
d             = Radius

;---- Setpoint -----
[string]
line          = 2
position      = 1
d             = Sollwert =

[value]
line          = 2
position      = 20
eparam        = 50
form          = 6.3
dimension     = mm
sign          = y

;---- Lower tolerance -----
[string]
line          = 3
position      = 1
d             = Untere Toleranz =

[value]
line          = 3
position      = 20
eparam        = 51
form          = 6.3
dimension     = mm
sign          = y

;---- Upper tolerance -----
[string]
line          = 4
position      = 1
d             = Obere toleranz =

[value]
line          = 4
position      = 20
eparam        = 52
form          = 6.3
dimension     = mm
sign          = y

```

```
;--- Actual value -----
```

```
[string]
```

```
line      = 5
position   = 1
d          = Istwert =
```

```
[value]
```

```
line      = 5
position   = 20
eparam     = 53
form       = 6.3
dimension  = mm
sign       = y
```

```
;--- Difference -----
```

```
[string]
```

```
line      = 6
position   = 1
d          = Differenz =
```

```
[value]
```

```
line      = 6
position   = 20
eparam     = 54
form       = 6.3
dimension  = mm
sign       = y
```

```
;-----
```

```
[string]
```

```
line      = 7
d          = *****
```

6.8 Array functions

6.8.1 Introduction to array functions:

Array functions can be used for handling two-dimensional numeric arrays. An array exists of rows and columns. A row number and a column number define an element (a value).

These functions allow you to interact with and manipulate arrays in various ways. Arrays are essential for storing, managing, and operating on sets of (a big number of) variables. For example: Storing of a great number of measuring positions and calculating of a centre position.

The advantage of the new features of the array functions is:

- Make it easier to define the format of two-dimensional arrays.
- Load array data from hard disk directly into CNC memory (during a program-run).
- Store complete array data on hard disk in one storing procedure and not one value after another.
- Manipulate array data directly in CNC memory.
- Store array data in a clear table format, so it can be easily examined.

Automatic Array Deletion

All arrays in memory, except the arrays which are loaded during start-up, will be deleted automatically from memory after: <Clear control>, <Cancel program>, <Cancel block> during EASYoperate, M30 and CNC system restart.

6.8.2 Overview array functions:

Function	Description	Source	Target
arrayNew()	Create a two-dimensional array in memory.		M
arraySave()	Store an array from memory on hard disk.	M	HD
arrayOpen()	Load an array from hard disk into memory.	HD	M
arrayExist()	Test the existence of an array on hard disk or in memory.	HD / M	
arraySize()	Determine the number of rows or columns in an array.	M	
ArrayFind()	Find data in an array.	M	
arrayWrite()	Add data to an array.		M
arrayRead()	Extract data from an array.	M	
arrayFilter()	Filter an array.	HD / M	M
arraySort()	Sort an array by column.	HD / M	M
arrayDelete()	Delete an array	HD / M	

Remarks:

- The third and fourth column describes the place where an array is stored ('HD' = hard disk, 'M' = memory).
- Several array functions are operating with arrays both on hard disk as well as in memory. Furthermore, due to a large amount of array data, it might be necessary to manipulate arrays directly on hard disk instead of loading the source data in memory first.
- The return values are stored in an E-parameter. E.g. E10=arrayExist().

6.8.2.1 arrayNew (format)

The goal of the function arrayNew() is to create a two-dimensional array in the memory of the CNC system.

<format> column names or number of columns
 If an array with column names must be created, these column names must be programmed between double quotes and delimited by the symbol '|'. If no column names are required, <format> must be programmed as a number.
 The length of each column name makes up the column width.

Returns: 0 if the array is not created.
 nnn an internal array identification number is given, when the array is created.
 E.g.: 1= first array, 2=second array, etc.

Example

This example creates an array in memory for tool data. The array contains three columns with the column names 'Tool', 'Length' and 'Radius'.

N1 E10=arrayNew(" Tool | Length | Radius ")

The return value (internal array identification number) is e.g. E10=xxx.

This example creates an array in memory, that contains three columns and no column names.

N1 E10=arrayNew(3)

6.8.2.2 arraySave (filename, internal array identification number)

The goal of the function arraySave() is to store an array from CNC memory on hard disk.

<filename> array name on hard disk.

The filename must be programmed between double quotes.

< internal array identification number > array name in CNC memory.

The array name must be programmed as a number or as an E-parameter (return value of arrayNew() or arrayOpen()).

Note: If the array <filename> already exists on hard disk, the contents of this array is overwritten.

Returns: 0 if the array is not save on hard disk.
 1 if the array is saved.

Format on hard disk.

The array written to the hard disk has the following format. This file can be edit with the editor.

For example an array with 3 columns. Each information is separated by"|".

```
[BEGIN]
Tool |Length.|Radius |
  1|  20.7|    5|
  2|   2.3|   5.7|
 10|  35,3|   5.8|
[END]
```

Example

This example saves an array file with tool data and with machine data.

N1 E1=xxx internal array identification number from arraynew

N2 E10=arraySave("\\Work\\Tool.arr", E1)

N3 E11=arraySave("\\Work\\Machine.arr", 2)

6.8.2.3 arrayOpen (filename)

The goal of the function arrayOpen() is to load an array from hard disk into the memory of the CNC system.
 <filename> array name on hard disk (entered between double quotes).

Returns: 0 if the array is not opened.
 nnn The array is loaded in memory.
 nnn is the unique internal identification number of the array (arrayNew).

Example

The following example opens an array file with tool data and with machine data. If these files are opened and successfully loaded, then arrays are created in memory.

N1 E10=arrayOpen("Work\Tool.arr")

The return value (internal array identification number) is for example E10=xxx.

N2 E11=arrayOpen("Work\Machine.arr")

6.8.2.4 arrayExist (name)

The goal of the function arrayExist() is to test the existence of an array on hard disk or in CNC memory.

<name> array name on hard disk or in memory.
 hard disk: string (between double quotes).
 memory: number or E-parameter (internal array identification number) (return value of arrayNew() or arrayOpen()).

Returns: 0 if the array does not exist.
 1 if the array exists.

Example

This example tests the existence of the array file 'Tool.arr' on hard disk.

N1 E10=arrayExist("Work\Tool.arr")

This example tests the existence of two arrays in memory.

N1 E1=9700 (internal array identification number)

N2 E10=arrayExist(E1)

N3 E11=arrayExist(9701)

6.8.2.5 arraySize (internal array identification number, rowcol)

The goal of the function arraySize() is to return the number of rows or columns in an array.

<internal array identification number > array name in memory.
 number or E-parameter (internal array identification number) (return value of arrayNew() or arrayOpen()).
 <rowcol> 1=determine the number of rows
 2=determine the number of columns.

Note: The number of rows in the array <name> equals

- The highest row number of a non-empty row, if this row is written by arrayWrite().
- The number of rows, if these rows are written by arrayOpen(), arraySort() or arrayFilter().

Returns: The number of rows in the array <name> if <rowcol> equals '1'.
 The number of columns in the array <name> if <rowcol> equals '2'.

Example

This example determines the number of columns in the array in memory.

N1 E10=arrayOpen("Work\Tool.arr")

N2 E11=arraySize(E10, 2)

6.8.2.6 arrayFind (internal array identification number, column, value)

The goal of the function arrayFind() is to return the number of the row in which the first occurrence of a value is found.

< internal array identification number > array name in memory.

<column> column number.

<value> value that must be found.

Returns

The row number in which the value <value> is found. If this value is not found in the programmed column, then the value '0' must be returned.

Example

The following array is stored in memory with internal array identification number stored in E40.

Id	Unbalance	Speed	Amplitude
10	100,000	25	0.00345
11	100,000	50	0.00862
20	200,000	25	0.00710
21	200,000	50	0.01992

N8 E41=arrayFind(E40, 1, 20) Find value= 20 in column= 1. The result E41= 3.

Remark: With arrayFilter an array with the desired value can be generated. On this manner the next row can be found.

6.8.2.7 arrayWrite (internal array identification number, row, column, value)

The goal of the function arrayWrite() is to add data to an array in CNC memory.

<internal array identification number > array name in memory.

number or E-parameter (internal array identification number) (return value of arrayNew() or arrayOpen()).

<row> row number.

<column> column number.

<value> value to be written in the array.

The array element(<row>,<column>) will be made empty,. If the <value> is programmed as '-999999999'

Returns: 0 if the value is not written in the array.
1 if the value is written.

Example

Tool	Length	Radius
1	20.7	5
2	42.3	5.7
10	35.5	5.8

This example loads the array in memory and after that it adds a complete new row to this array in memory.

N1 E10=arrayOpen("Work\Tool.arr") E10= internal array identification number

N2 E20=arrayWrite(E10, 4, 1, 11)

N3 E21=arrayWrite(E10, 4, 2, 46.0)

N4 E22=arrayWrite(E10, 4, 3, 10.6)

Tool	Length	Radius
1	20.7	5
2	42.3	5.7
10	35.5	5.8
11	46.0	10.6

Note that the changed array must be saved to harddisk with arraySave.

6.8.2.8 arrayRead (internal array identification number, row, column)

The goal of the function `arrayRead()` is to extract data from an array in CNC memory and store it in an E-parameter.

< internal array identification number > array name in memory.

number or E-parameter (internal array identification number) (return value of arrayNew() or arrayOpen()).

<row> row number.

<column> column number.

Returns

The value in array element(<row>,<column>). If this element in the array is empty, then the value '999999999' must be returned.

Example _____

Tool	Length	Radius
1	20.7	5
2	42.3	5.7
10	35.5	5.8

This example first loads the array in memory. After that it reads the element in the third row of the first column from this array in memory.

N1 E10=arrayOpen("\\Work\\Tool.arr") E10= internal array identification number

```
N2 E20=arrayRead(E10, 3, 1)
```

Parameter E20 contains now the value 10

6.8.2.9 arrayFilter (name, column, criteria)

The goal of the function `arrayFilter()` is to return a filtered array. This filtered array consists of the rows that contains the value to filter on.

<name> array name on hard disk or in memory.

hard disk: string (between double quotes).

memory: number or E-parameter (return value of arrayNew() or arrayOpen()).

<column> column number.

<criteria> criteria expression be used for filtering.

For the parameter <criteria>, all expressions are allowed, which are also allowed for DIN programming. An example is the following expression: ($\leq \sin(90)$).

It returns a filtered array with all values smaller than and equal to $\sin(90)$.

Returns: 0 if the array is not filtered.

0 if the array is not filtered.

nnn internal array identification number

Example

Unbalance	Speed	Amplitude
100000	25	0.00345
100000	50	0.00862
200000	25	0.00710

This example filters the first column of the array on hard disk and stores the result in memory.

N1 E10=arrayFilter("\Work\Balance.arr", 1, 100000)	E10= internal array identification number
--	---

Unbalance	Speed	Amplitude
100000	25	0.00345
100000	50	0.00862

6.8.2.10 **arraySort (name, column, order)**

The goal of the function arraySort() is to return a column sorted array.

<name> array name on hard disk or in memory.
 hard disk: string (between double quotes).
 memory: number or E-parameter (internal array identification number) (return value of arrayNew() or arrayOpen()).

<column> column number.

<order> sort order; 1=ascending and 2=descending

Note: If the non-sorted array contains empty rows, the number of rows in the sorted array must be less than the number of rows in the non-sorted array.

Returns: 0 if the array is not sorted.
 nnn internal array identification number.

Example

Unbalance	Speed	Amplitude
100000	25	0.00345
100000	50	0.00862
200000	25	0.00710

This example sorts the third column of the array on hard disk ascending and stores the result in memory.

N1 E10=arraySort("\Work\Balance.arr", 3, 1) E10= internal array identification number

Unbalance	Speed	Amplitude
100000	25	0.00345
200000	25	0.00710
100000	50	0.00862

6.8.2.11 **arrayDelete (name)**

The goal of the function arrayDelete() is to delete an array from hard disk or from CNC memory.

<name> array name on hard disk or in memory.
 hard disk: string (between double quotes).
 memory: number or E-parameter (internal array identification number) (return value of arrayNew() or arrayOpen()).

Returns: 0 if the array is not deleted.
 1 if the array is successfully deleted.

Example

This example deletes an array from hard disk.

N1 E10=arrayDelete("\Work\Tool.arr")

This example deletes an array from memory.

N1 E10=arrayOpen("\Work\Tool.arr") E10= internal array identification number

N2 E11=arrayDelete(E10)

6.8.3 Method with Configuration file (previous versions)

In the previous versions the following a restricted possibility was implemented.
It is advisable to use only the new functionality.

Configuration file

Configuration files are required to describe how and where to write or read.

These configuration files are saved on the hard disk:

D:\STARTUP\CYCLES\ARRnnnnn.CFG

nnnnn is the file number from 1 to 89999.

File to define an array and fill it with basic settings

An array is defined with a configuration file.

This is activated when the system is started.

A maximum of 10 arrays can be defined.

End users can define files themselves.

The maximum size for all arrays together is 5000 elements.

Description of an array configuration file:

```
;Comments start with ';'
;
;Sections:
[element]
;row   =      row number      where row number = [1|...|9999]
;col   =      column number   where column number = [1|...|9999]
;      row * column <= 5000
;val   =      value           where value = real number (double)
;
```

Filling a configuration file

The configuration file can be filled with values (arrays). These arrays can be read (arrayRead) during execution like E-parameters. There is no function to write values in the array during execution

Example: Array configuration file:

ARRnnnnn.CFG

```
[element]
row   =      1
col   =      1
val   =      0           ; element (1,1).=.0

[element]
row   =      3
col   =      66
val   =      397.01      ; element (3,66) = 397.01

[element]
row   =      9999        ;maximum row size
col   =      9999
val   =      -123456789.123456789
```

arrayread (arraynumber, row, column)

arraynumber is the number of the array. Every array has its own configuration file. Arraynumber between 1 and 89999.

Row is the row number in the array that is to be read. Row between 1 and 999999.

Column is the position in the row of the array that is to be read. Column between 1 and 999999.

Fixed arrays can be read with the arrayread function. The arrays are filled from a configuration file D:\STARTUP\CYCLES\ARRnnnnn.CFG).

Empty 'elements' in the array have the value <-999999999>.

Example arrayread
E300 = arrayread(100,1,2)
E300 has the value of array 100, row 1, column 2.

7. Tool measuring cycles for laser measuring

7.1 General remarks for laser measuring

Laser measuring is extended with the following G-functions:

G951 Calibration.	replaced G600
G953 Measure tool length	replaced G601
G954 Measure length, radius	replaced G602
G955 Cutter control shank	replaced G603
G956 Tool breakage control	replaced G604
G957 Cutter control shape.	
G958 Tool setting length, radius, corner radius.	

For the explanation of these G-functions, see: Manual Blum.

For laser measuring of turning tools: see G615 in chapter: Turning.

For laser measuring of temperature compensation: see G642 in chapter: Measuring cycles.

Availability

The machine and MillPlus **IT** must be prepared by the machine manufacturer for the measuring instrument. If not all the G functions described here are available on your machine, consult your machine handbook.

Programming

Before calling one of the G600-G609 functions a M24 (active measuring system) must be programmed, so that the measuring system is set in the measuring position.

After measuring a M28 (deactive measuring system) must be programmed, so that the measuring system is retract.

Rotary axes, if any, are not taken into account or positioned.

The free machining plane G7 must not be active

Machine constants

The G function and associated machine constants are activated via the following machine constants.

MC261 >0: Measurement cycle functions

MC254 >0: Measure tool

MC840 =1: Measuring probe present

MC854 =1: Tool measuring instrument type (0=none, 1=laser, 2=TT130)

MC350 Probe position 1st axis negative μm

MC351 Probe position 1st axis positive μm

MC352 Probe position 2nd axis negative μm

MC353 Probe position 2nd axis positive μm

MC354 Probe position 3rd axis negative μm

MC355 Probe position 3rd axis positive μm

MC350 to MC355 are operator machine constants and are detected when calibrating.

MC356 Axis number for radial measuring: 1=X, 2=Y, 3=Z

MC357 Tool axis number for measuring: 1=X, 2=Y, 3=Z

MC358 Measure: 3rd axis 0=no, 1=yes

MC359 Radial probe contact side: -1=neg, 0=aut, 1=pos

GENERAL REMARKS FOR LASER MEASURING

MC360 -- MC369 are intended for a second laser measurement device in another work area or an adapter spindle. The area used is determined by the IPLC.

MC370 Maximum tool radius μm

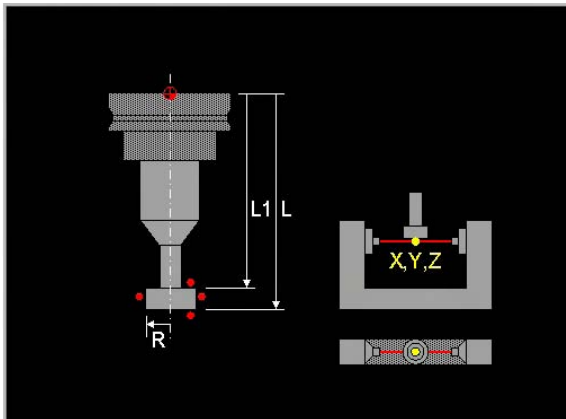
MC371 Maximum tool length μm

MC372 Clear space under laser beam μm

MC373 free space behind the laser beam in μm

7.2 G600 Laser: Calibration

To determine the position of the laser measuring instrument and store this position value in the machine constants provided.



```
G   Laser: Calibration
S   Speed
X   Measuring point
Y   Measuring point
Z   Measuring point
I1= Swing determination 0=no 1=yes
```

Notes and application

Determining concentricity error (I1=)

Use address I1 to specify whether the concentricity error is to be measured and saved in the tool table against the calibration tool. It is obligated, that the concentricity error should be determined once using a clean calibration stylus.

I1= 0 Do not determine concentricity error (basic setting)
 1 Determine the concentricity error

The radial concentricity error is written to the tool memory under R4=.

The axial concentricity error is written to the tool memory under L4= and the length L is reduced by the L4 value. The sum L+L4 remains constant.

Speed

S = Speed (recommended value S3000)

Coolant will be thrown off by clockwise-anticlockwise-clockwise rotation.

The spindle is switched off with M5 at the end of the cycle.

Calibration stylus, tool memory addresses

The dimensions of the calibration mandrel are entered in the tool memory.

L Length of calibration mandrel (underside of cylindrical portion)

R Radius

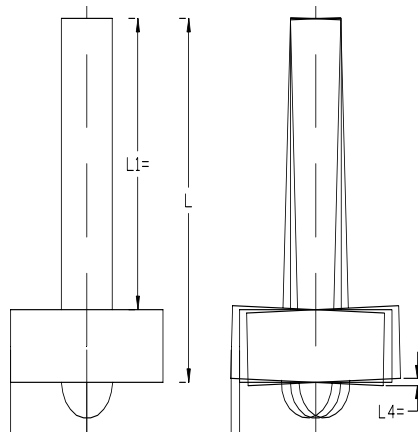
L1= Second length (top of cylindrical portion)

The second length L1= is not entered if a fully cylindrical calibration pin is used. In this case, only the top of the laser beam is calibrated.

The concentricity errors R4 and L4 of the calibration stylus are written to the tool memory by the calibration cycle.

R4= Radial concentricity error of calibration stylus.

L4= Axial concentricity error of calibration stylus.



Definition of calibration tool in tool memory.

Position of measuring unit

X,Y,Z is the global position (to within +/- 5 mm) of the measuring instrument relative to the machine zero point.

If X,Y or Z are not entered, the calibrated positions from the machine constants are used.

When determining the position of the measuring unit for the calibration, the centre of the bottom edge of the pin (dimension L) must be set in the light beam (+/- 5 mm).

At calibration, the exact position of the measuring instrument is measured and stored in MC350-MC355. The stored values are relative to the reference point of the machine.

- Zero point offset must not be active if X,Y or Z are entered.
- A calibration tool must be chosen. T0 is not permitted.

Example

Example 1 Calibrating laser measuring instrument and storing the position value in the E parameter.
N... G600 X300 Y500 Z600 S3000

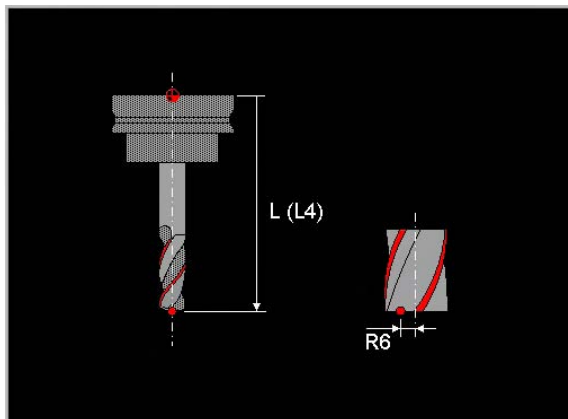
Example 2 Calibration of laser measuring unit, determining concentricity error.
N... G600 X300 Y500 Z600 I1=1 S3000

Concentricity errors L4 and R4 are saved in the tool table, length is matched (I1=1).

The exact X, Y and Z positions are saved in the machine constants.

7.3 G601 Laser: Measure tool length

To measure the length of centric tools.



```
G Laser: Measure tool length
S Speed
I1= Measuring side 0=lower 1=upper
```

Notes and application

Tool edge (I1=): The lower edge or the upper edge of the tool can be measured.
I1=0 measure lower edge (basic setting) and I1=1 measure upper edge

Speed (S) (recommended value S3000)

If the spindle is not first switched off (M5 or M19), then:

- Coolant will be thrown off by clockwise-anticlockwise-clockwise rotation.
- The spindle is switched off with M5 at the end of the cycle.

If the spindle is already switched off (M3 or M4), change of direction or spindle stop does not occur at the end of the cycle

The following addresses of the tool memory are used:

L Tool length
L4= Allowance length
L5= Length tolerance
R6= Radius position for measuring length.
E Tool status

Actions

Check (E=1): The measured difference is added to L4 in the tool table.

Measure (E=0 or no value):

When the first measurement is made, the tool length is overwritten, and allowance L4=0 and tool status E=1 are set.

- Speed-dependent measurement feed is calculated by the cycle.

Tool status

If the tolerance is exceeded the tool status E=1 is set.

If the tool status is E=1 at the start of the cycle, the cycle is skipped.

Length measurement

If the tool radius is greater than MC373 and R6 is not programmed, the length is measured eccentrically.

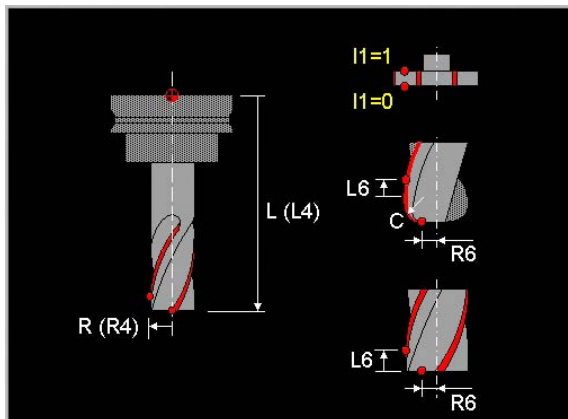
If R6 is programmed and $R - R6 > MC373$, an error message is issued

Working method by length measuring of a upper edge (I1=1) of an unknown tool:

- First the lower edge is measured in the middle.
- Then the tool move sideways to the radius position (R6=)
- The tool is positioned 2 mm above the clear space under the laser beam.
- The upper edge is measured pushing upwards.

7.4 G602 Laser: Measure length and radius

To measure the length and radius of acentric tools with laser measuring instrument



```
G   Laser: Measure length and radius
S   Speed
I1= Measuring side 0=lower 1=upper
I2= Double-sided meas. 0=no 1=yes
```

Notes and application

Selecting the tool edge (I1=)

The lower edge or the upper edge of the tool can be measured.

I1= 0 measure lower edge (basic setting)
 1 measure upper edge

Selecting measurement of one or both edges (I2=)

One or both edges of the tool can be measured.

I2= 0 measure one side (basic setting)
 1 measure both sides

When measuring both edges, temperature errors and tool obliquity have no influence on the measured radius.

Speed

S = Speed (recommended value S3000)

If the spindle is not first switched off (M5 or M19), then:

- Coolant will be thrown off by clockwise-anticlockwise-clockwise rotation.
- The spindle is switched off with M5 at the end of the cycle.

If the spindle is already switched off (M3 or M4), change of direction or spindle stop does not occur at the end of the cycle

Addresses of tool memory

The following addresses of the tool memory are used:

L Tool length
 L4= Length allowance
 L5= Length tolerance
 R Tool radius
 R4= Allowance radius
 R5= Radius tolerance
 L6= Position above the tool tip for true running check
 R6= Radius position for length measurement
 Q4= Number of teeth
 E Tool status
 C Corner radius

Actions

Check (E=1)

The measured deviation is added to L4 and R4 in the tool table.

Measure (E=0 or no value)

When the first measurement is made, the tool length and radius are overwritten, and allowance L4 and R4 =0 and tool status E=1 are set.

- Speed-dependent measurement feed is calculated by the cycle.

Tool status

If the tolerance is exceeded the tool status E-1 is set.

If the tool status is E=1 at the start of the cycle, the cycle is skipped.

Length measurement

- If the tool radius is greater than MC373 and R6 is not programmed, the length is measured eccentrically.
- If R6 is programmed and $R-R6 > MC373$, an error message is issued.

Radius measurement

- If L6=0 no radius measurement is carried out.
- If L6 is greater than MC372, an error message is issued.

True running check

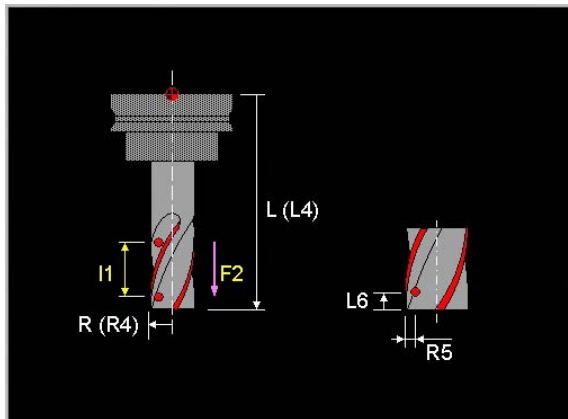
- If $Q4 > 0$ (number of teeth from the tool table), a true running check is carried out after the radius measurement.
- The true running check is carried out at a calculated rpm.
- The speed superimpose switch is not active.

Working method by length measuring of a upper edge (I1=1) of an unknown tool:

- First the lower edge is measured in the middle.
- Then the tool move sideways to the radius position (R6=)
- The tool is positioned 2 mm above the clear space under the laser beam.
- The upper edge is measured pushing upwards.

7.5 G603 Laser: Check of individual edge

To monitor the lower part (inspection height) of the tool with a laser measuring instrument.



G Check of individual edge
F2= Scanning feed
I1= Tool travelling distance

Notes and application

Addresses of tool memory

The following addresses of the tool memory are used:

L	Tool length
L4=	Length allowance
R	Tool radius
R4=	Radius allowance
R5=	Radius tolerance
L6=	Position above the tool tip for true running check
Q4=	Number of teeth
E	Tool status

Tool status

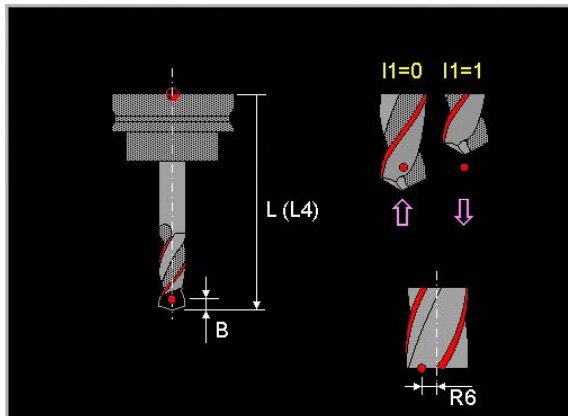
If the tolerance is exceeded the tool status E-1 is set.

If the tool status is E=1 at the start of the cycle, the cycle is skipped.

- If I1=0, only a true running check is carried out.
- The edge check is carried out at a calculated speed.
- The speed superimpose switch is not active.
- Maximum error is laid down via R5.
- If I1+L6 is greater than MC372, an error message is issued

7.6 G604 Laser: Tool breakage control

Tool breakage check



```
G   Laser: Tool breakage control
S   Speed
I1=  Meas. direction 0=pull 1=push
I2=  0=Error/Pallette 1=no error
O1=  E-param. number for tool status
```

Notes and use

Measuring direction (I1=)

The measuring direction can be pushing or pulling.

I1= 0 pulling (basic setting)
1 pushing

The fast pulling measurement is preferred, but tools with pronounced concave grinding must be measured pushing, as otherwise the hollow grinding will be detected as a break.

Error evaluation (I2=)

If a break is detected, various actions can follow:

I2= 0 error message or reject pallet (basic setting)
1 no error message

If I2=0 is selected, function M105 (tool break detected) is issued in the case of tool breakage. The IPLC switches the laser off and the controller issues an error message.

If, however, a pallet system is present, the pallet is rejected if possible, the current program is interrupted and a new pallet is brought in.

If I2=1 is selected, no error message is issued on tool breakage. Every action must be programmed in the part program. To achieve this, the tool status (value E from the tool memory) can be written directly to an E parameter. See address O1.

Tool status output to e parameter (O1=)

The tool status (definition E in the tool memory) is written to the specified E parameter. Based on this parameter, the program can determine whether a tool breakage has been detected (status -4). This is meaningful, if the error message has been switched off with I2=1.

Speed

S = speed (recommended value S3000)

If the spindle is not first switched off (M5 or M19), then:

- Spindle is switched on clockwise (M3).
- The spindle is switched off with M5 at the end of the cycle.

If the spindle is already switched off (M3 or M4), spindle stop does not occur at the end of the cycle.

Addresses of tool memory

The following addresses of the tool memory are used:

L	Tool length
L4=	Length allowance
R	Tool radius
R4=	Radius allowance
B	Breakage tolerance in mm (also in inch mode)
R6=	Radius position for breakage check
E	Tool status

Tool status

- When the breakage tolerance is exceeded, tool status E-4 is set and in addition an alarm is issued.
- Even if the tool status is E=1 at the start of the cycle, the breakage check is carried out
- The basic setting for tolerance B is entered in MC33. Only 1 or 2 mm is possible. The setting of MC133 is in mm even in inches mode.
- Breakage monitoring must be turned on by means of MC32.

Breakage measurement

If the tool radius is greater than MC373 and R6 is not programmed, the length is measured eccentrically.

If R6 is programmed and $R - R6 > MC373$, an error message is issued

8. Measuring system "Table-Probe" (TT)

8.1 General notes measuring system "Table-Probe" (TT)

Remark: TT means "Table Probe", for example TT130 or a similar instrument.

Availability

The machine manufacturer for the measuring instrument must prepare the machine and MillPlus **IT**. If not all the G functions described here are available on your machine, consult your machine handbook.

Programming

Before calling one of the G600-G609 functions a M24 (active measuring system) must be programmed, so that the measuring system is set in the measuring position.

After measuring a M28 (deactivate measuring system) must be programmed, so that the measuring system is retract.

Machine constants

The G function and associated machine constants are activated via the following machine constants.

MC 261 >0	measurement cycle functions
MC 254 >0	measure tool
MC 840 =1	measurement probe present
MC 854 =2	tool measuring instrument type (0=none, 1=laser, 2=TT)

MC 350	Probe position 1st axis μm
MC 352	Probe position 2nd axis μm
MC 354	Probe position 3rd axis μm

Coordinates of the TT stylus centre point relative to the machine zero point G51 and G53 (-max - +max μm)

After calibration the exact positions is written in MC350 – Mc355.

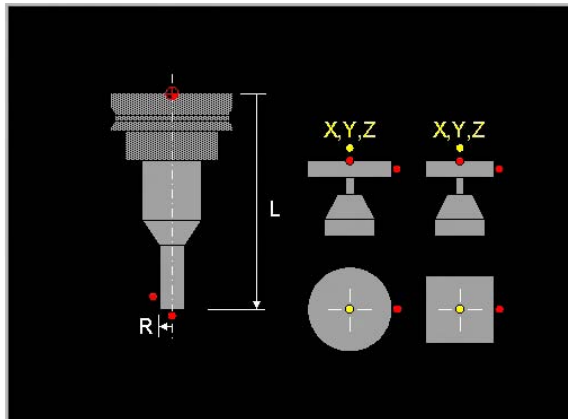
MC 356	axis number for radial measurement: 1=X, 2=Y, 3=Z
MC 357	tool axis number for measuring: 1=X, 2=Y, 3=Z
MC 358	measuring: 3rd axis 0=no, 1=yes
MC 359	radial probe contact side: -1=negative, 0=automatic, 1=positive

MC 360 -- MC 369 are for the second laser measurement system a second work area or an attachment spindle. Which area will be used, is determinates by the IPCL.

MC 392	maximum permitted measurement error for tool measurement with rotating tool (2 - 1000 μm)
MC 394	probe measuring feed with tool measurement with non-rotating tool (10 - 3000 mm/min)
MC 395	distance from tool underside to stylus top for tool radius measurement (1 - 100000 μm)
MC 396	diameter or side length of the stylus of the TT. (1 - 100000 μm)
MC 397	safety zone around the stylus of the TT for pre-positioning. (1 - 10000 μm)
MC 398	rapid in measuring cycle for TT. (10 - 10000 mm/min)
MC 399	maximum permitted rotational speed at tool edge (1 - 120 m/min).

8.2 G606 TT: Calibration

To determine the position of the measuring instrument and store this position value in the machine constants provided.



```
G  TT130: Calibration
X  Measuring point
Y  Measuring point
Z  Measuring point
```

Notes and use

Calibration tool

Before you calibrate, you must enter the exact radius and the exact length of the calibration tool in the tool table.

Sequence

The calibration process runs automatically. MillPlus **IT** also determines the centre offset of the calibration tool automatically. For this, MillPlus **IT** rotates the spindle after half of the calibration cycle by 180°. As a calibration tool, use an exactly cylindrical part, e.g. a cylindrical pin. MillPlus **IT** stores the calibration values in the machine constants and takes them into account in the subsequent tool measurements.

In MC 350, MC 352, MC 354 the position of the TT in the work area of the machine must be stipulated.

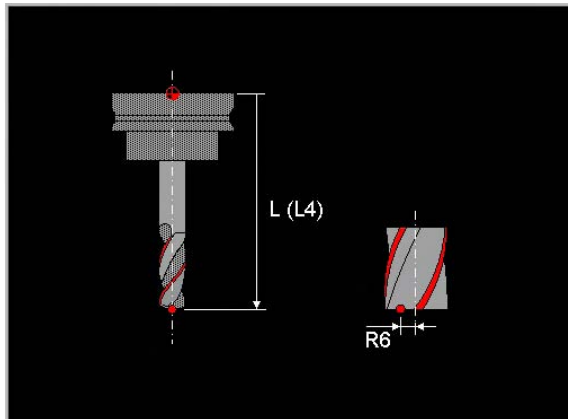
If you change one of MC 350, MC 352, MC 354, you must re-calibrate.

Position

Input in the X, Y and Z-axes, the position in which the possibility of collision with workpieces or clamping fixtures is excluded. If the position height input is so small that the tip of the tool would be below the plate surface, MillPlus **IT** positions the calibration tool above the plate non-automatically.

8.3 G607 TT: Measuring tool length

To measure the tool length.



```
G   TT130: Measure tool length
I2= All teeth  0=no 1=yes
I1= Clearance
```

Notes and use

Tool length and radius

Before you measure tools for the first time, enter the approximate radius (R10), the approximate length (L100), the number of cuts (Q4=4) and the cutting direction (I2=0) of the tool to be used in the tool table.

Addresses of the tool memory

The following addresses of the tool memory are used:

L	tool length
L4=	length allowance
L5=	length wear tolerance
R	tool radius
R4=	radius allowance
R6=	measurement offset radius
E	tool status

Sequence

The tool length can be determined in three different ways:

- 1 If the tool diameter is greater than the diameter of the measurement surface of the TT, measure with tool rotating.
- 2 If the tool diameter is smaller than the diameter of the measurement surface of the TT or if you determine the length of drills or radius cutters, measure with tool stationary.
- 3 With the parameter I2=1 all teeth are measured. The measurement is carried out with stationary spindle. The greatest tooth length is entered in the tool table.

Measuring with tool rotating

To determine the longest edge, the tool to be measured is offset to the probe system centre point and moved, rotating, onto the measurement surface of the TT. Program the offset in the tool table under tool offset; radius (R).

Measuring with tool stationary (e.g. for drills).

The tool to be measured is moved to be concentrically above the measurement surface. Then it travels with the spindle stationary onto the measurement surface of the TT. For this measurement enter the tool offset: radius (R6=0) in the tool table.

Individual edge measurement

MillPlus **IT** pre-positions the tool to be measured to the side of the probe. The end face of the tool is then located below the probe top as laid down in MC 395. In the tool table, you can stipulate an additional offset under tool offset; length (L). MillPlus **IT** applies the probe radial with the tool rotating, to determine the start angle for the individual edge measurement. It then measures the length of all edges by changing the spindle orientation. For this measurement, select the Softkey all teeth.

Measure tool (E=0 or no value)

During the initial measurement, MillPlus **IT** overwrites the tool radius (R10 with R10.012) and the tool length (L100 with L99.456) in the tool memory and sets the oversizes R4 and L4 = 0.

Check tool (E=1)

During the initial measurement, MillPlus **IT** overwrites the tool length L in the tool memory and sets the oversize L4=0. In the event that you are checking a tool, the actual length measured is compared with tool length L extracted from the tool table. MillPlus **IT** calculates the mathematically correct variance and enters this as the oversize L4 in the tool table. If this oversize is greater than the permissible wear or breakage tolerance for the tool length, then a fault report is made.

Safe height (I1=):

Enter a position in the spindle axis, by means of parameters from the entry dialog (I1 = safety distance), such that a crash with pieces of work or their supporting holders is excluded. The safe height refers to the reference point for the active piece of work. If the safe height entered is so small that the tool tip would lie below the top surface of the plate, MillPlus **IT** does not automatically place the tool over the plate (security zone from MC397)

Cut measurement (I2=):

switch on or off individual cut measurement (Parameter I2=)
With I2=0 or no value, individual edge measurement is carried out.

Difference EASYoperate and DIN.

In EASYoperate is parameter edge measurement (I2=) replaced by a Softkey "all Teeth".

Stationary spindle

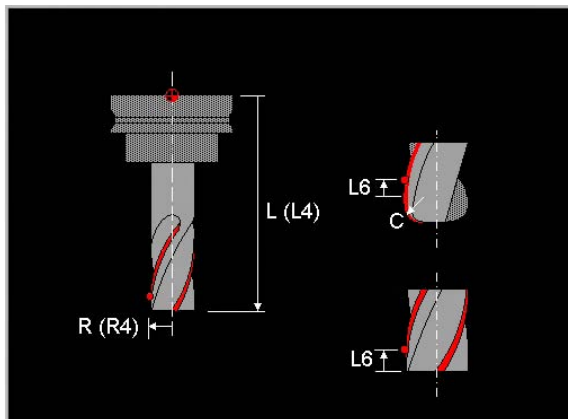
MillPlus **IT** uses the probe measuring feed from MC 394 for the measurement with stationary spindle.

Calculation of the spindle Speed

When measuring with a tool, MillPlus **IT** calculates the spindle speed and the probe measuring feed automatically.

8.4 G608 TT: Measuring tool radius

To measure the tool radius.



```
G   TT130: Measure tool radius
I2= All teeth  0=no 1=yes
I1= Clearance
```

Notes and use

Tool length and radius

Before you measure tools for the first time, enter the approximate radius (R10), the approximate length (L100), the number of cuts (Q4=4) and the cutting direction (I2=0) of the tool to be used in the tool table.

Addresses of the tool memory

The following addresses of the tool memory are used:

L	tool length
L4=	length allowance
R	tool radius
R4=	radius allowance
R5=	radius wear tolerance
E	tool status

Measure tool (E=0 or no value)

During the initial measurement, MillPlus **IT** overwrites the tool radius (R10 with R10.012) and the tool length (L100 with L99.456) in the tool memory and sets the oversizes R4 and L4 = 0.

Measurement sequence

You can determine the tool radius in two ways:

- 1) Measurement with rotating tool
- 2) Measurement with rotating tool and subsequent individual edge measurement

With individual edge measurement, the radius is first measured roughly and the position of the largest tooth determined. After that, the other teeth are measured.

MillPlus **IT** pre-positions the tool to be measured to the side of the probe. The milling cutter end face is then below the top of the probe, as laid down in MC 395. MillPlus **IT** applies probe measuring radial with rotating tool. If an individual edge measurement is also to be carried out, the radii of all edges are measured by means of spindle orientation.

Check tool (E=1)

If you check a tool, the measured radius is compared with the tool radius R from the tool table. MillPlus **IT** calculates the difference with correct sign and enters this as allowance R4 in the tool table. If the allowance is greater than the permitted wear (R5=) or breakage tolerance for the tool radius, an error message is output.

Clearance (I1=)

Enter a position in the spindle axis, by means of parameters from the entry dialog (I1 = safety distance), such that a crash with pieces of work or their supporting holders is excluded. The safe height refers to the active workpiece reference point. If the safe height entered is so small that the tool tip would lie below the top surface of the plate, MillPlus **IT** does not automatically place the tool over the plate (security zone from MC397)

Edge measurement (I2=)

With parameter I2=1 all teeth are measured.

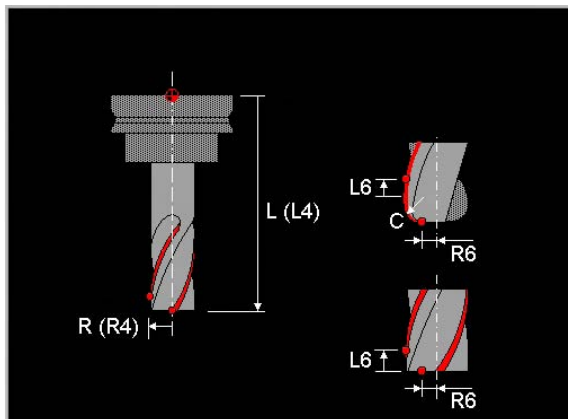
With I2=0 or no value, an individual edge measurement is carried out.

Difference EASYoperate and DIN.

In EASYoperate is parameter edge measurement (I2=) replaced by a Softkey "all Teeth".

8.5 G609 TT: Measuring length and radius

To measure tool length and radius.



```
G    TT130: Measure length and radius
I2=  All teeth  0=no 1=yes
I1=  Clearance
```

Notes and use

Tool length and radius

Before you measure tools for the first time, enter the approximate radius (R10), the approximate length (L100), the number of cuts (Q4=4) and the cutting direction (I2=0) of the tool to be used in the tool table.

Addresses of the tool memory

The following tool memory addresses are used:

L	tool length
L4=	length allowance
L5=	length wear tolerance
R	tool radius
R4=	radius allowance
R5=	radius wear tolerance
E	tool status

Measurement sequence

MillPlus **IT** measures the tool according to a fixed, programmed sequence. First the tool radius and then the tool length are measured.

You can determine the tool radius in two ways:

- 1) Measurement with rotating tool
- 2) Measurement with rotating tool and subsequent individual edge measurement

Measure tool (E=0 or no value)

The function is especially suitable for the first measurement of tools since, compared with the individual measurement of length and radius, there is a considerable time advantage.

With the first measurement, MillPlus **IT** overwrite the tool radius R and tool length L in the tool memory and sets the allowance R4 and L4 = 0.

Check tool (E=1)

If you check a tool, the measured tool data are compared with the tool data from the tool table. MillPlus **IT** calculates the differences with correct signs and enters these as allowance R4 and L4 in the tool table. If an allowance is greater than the permitted wear (L5= and R5=) or breakage tolerance for the tool radius, an error message is output.

Clearance (I1=)

The clearance (I1=) in the direction of the spindle axis, excluded the possibility of a collision with workpieces or clamping fixtures. The clearance relates to the top of the measuring device. Default I1=MC397

Edge measurement (I2=)

With parameter I2=1 all teeth are measured.

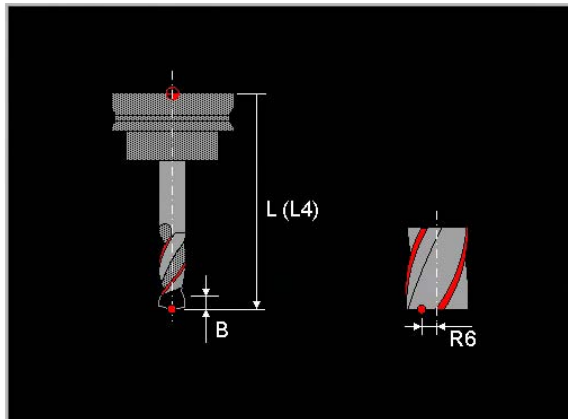
With I2=0 or no value, an individual edge measurement is carried out.

Difference EASYoperate and DIN.

In EASYoperate is parameter edge measurement (I2=) replaced by a Softkey "all Teeth".

8.6 G610 TT: Tool breakage control

Monitoring tool length. Mainly used for monitoring tools that are liable to break, such as drills. The measured wear is not corrected.



```
G   TT130: Tool breakage control
I2= All teeth  0=no 1=yes
I1= Clearance
I3= 0=Error/Pallet 1=no error
O1= E-param. number for tool status
```

Hinweise und Verwendung

Tool data

Tool data must be entered in the tool table beforehand. No measurement is done where the tool status is -1 or -4.

Addresses of tool memory

The following addresses of the tool memory are used:

- L Tool length
- L4= Length allowance
- R6= Radius position for breakage check
- B Breakage tolerance in mm (also in inch mode)
- E Tool status

For individual cutting measurement:

- R Tool radius
- R4= Radius allowance
- L6= Length position for breakage check

Differences between EASYoperate and DIN:

This function is not available in EASYoperate.

Sequence

Tool breakage, like tool length, can be determined in three different ways.

- 1 If the tool diameter is greater than the measuring surface of the TT, then measure with the tool rotating.
- 2 If the tool diameter is less than the measuring surface of the TT, then measure with the tool stationary. The same applies if you wish to determine the length of drills or radiusing mills.
- 3 All teeth are measured using parameter I2=1. This measurement is carried out with the spindle stationary.

Measuring with a rotating tool

The tool to be measured is offset to the sampling system centre and brought to the TT measuring surface while rotating. You must program the offset in the tool table under tool offset radius (R6=).

Measurement with stationary tool (e.g. drill):

The tool to be measured is centred above the measuring surface. Then it advances with a stationary spindle to the TT measuring surface. For this measurement, enter the tool offset radius (R6=0) in the tool table.

Individual cutting measurement

The MillPlus **IT** positions the tool to be measured at the side of the probe. The front surface of the tool is then below the top edge of the probe, as laid down in MC395. You can define an additional offset in the tool table under tool offset length (L6=). MillPlus **IT** scans radially with the tool rotating in order to determine the starting angle for the individual cutting measurement. It then measures the length of all cuts by changing the spindle orientation. For this measurement, you select I2=1"

Safety distance (I1=)

The setup clearance (I1=) in the direction of the spindle axis must be sufficient to prevent any collision with the workpiece or clamping devices. The setup clearance is with respect to the top edge of the stylus. Basic setting I1=MC397

Cutting measurement (I2=)

If I2=1 an individual cutting measurement is carried out.

If I2=0 or no value, individual cutting measurement is deselected.

Error evaluation (I3=)

If a break is detected, various actions can follow:

I3= 0 error message or reject pallet (basic setting)

I3= 1 no error message

If I3=0 is selected, function M105 (tool break detected) is issued in the case of tool breakage. The IPLC switches the TT off and the controller issues an error message.

If, however, a pallet system is present, the pallet is rejected if possible, the current program is interrupted and a new pallet is brought in.

If I3=1 is selected, no error message is issued on tool breakage. Every action must be programmed in the part program. To achieve this, the tool status (value E from the tool memory) can be written directly to an E parameter. See address O1.

Tool status output to E parameter (O1=)

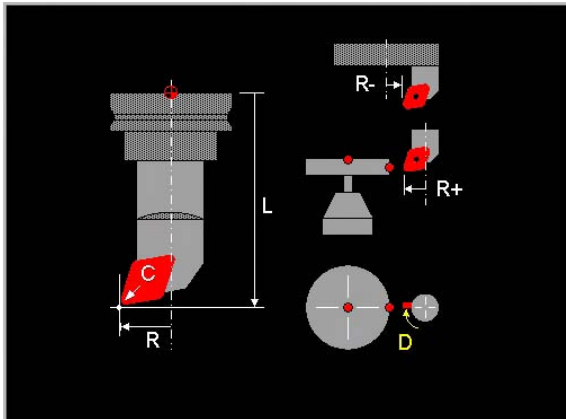
The tool status (definition E in the tool memory) is written to the specified E parameter. Based on this parameter, the program can determine whether a tool breakage has been detected (status 4). This is only meaningful if the error message has been switched off with I3=1.

Stationary spindle

For measurement with a stationary spindle, MillPlus **IT** uses the scanning feed from MC394.

See G607 for calculation of the spindle speed or scanning feed.

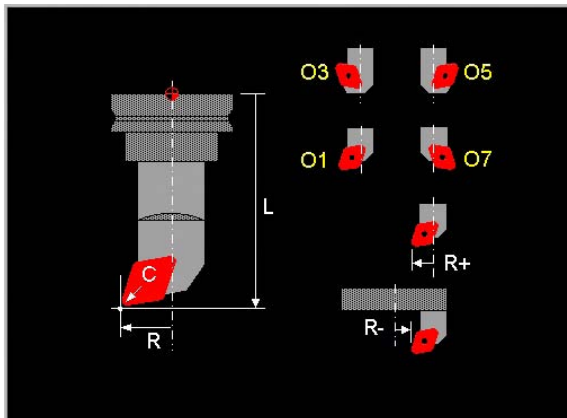
8.7 G611 TT: Measuring turning tools



G TT130: Turning tool measurement
 D Orientation angle tool tip
 I1= Clearance
 I4= Measuring: 0=L+R 1=L 2=R

Refer to Chapter "Turning mode".

8.8 G615 Laser: Measuring turning tools



```
G Laser: Turning tool measurement
D Orientation angle tool tip
O Tool orientation
```

Refer to Chapter "Turning mode".

9. Measuring cycles

9.1 Introduction to measuring cycles

Measuring cycles in the main plane:

G620	Angle measurement
G621	Position measurement
G622	Corner outside measurement
G623	Corner inside measurement
G626	Datum outside rectangle
G627	Datum inside rectangle
G628	Circle measurement outside
G629	Circle measurement inside

Special measuring cycles:

G631	Measure the inclination of a plane (G7)
G633	Angle measurement 2 holes
G634	Measurement center 4 holes
G640	Rotary table center offset.
G642	Laser: temperature compensation

Comments

Comments are not allowed in a block with a machining cycle.

Results of activating a measuring cycle:

- G91 is deactivated.
- Radius correction is deactivated (G40 is active)
- Scaling with G72 is deactivated
- L and R in G39 are zeroed.

	G17	G18	G19
Main axis	X	X	Y
Secondary axis	Y	Z	Z
Machining plane	XY	XZ	YZ
Tool axis	Z	Y	X or -X (G66/G67)

In some cycles the direction of measurement is determined by the address (I1=).

Zero point

Measured values (I5>0) can be stored in the zero offset table where an offset is currently active and/or in an E parameter.

Restriction with G7: measured values can only be written in an E parameter. (I5= must only be zero).

Differences between EASYoperate and DIN/ISO

Certain addresses are not available in EASYoperate. The measured values are displayed in a window.

Comments

Comments are not allowed in a block with a machining cycle.

Results of activating a measuring cycle:

- G91 is deactivated.
- Radius correction is deactivated (G40 is active)
- Scaling with G72 is deactivated
- L and R in G39 are zeroed.

Machine constants that are important for measuring cycles

MC261 >0: Measuring cycle functions active
 MC312 =1: Free machining plane active (G631)
 MC840 =1: Measuring probe present
 MC843: Measuring feed
 MC846 >0: Angle of orientation of measuring probe
 MC849 : Probe 1. angle of orientation

Functions that are not allowed when a measuring cycle is called.

G36, rotations (B4=) in G92/G93 and G182.
 G7 must not be active if the measured values are stored in zero point offset (I5>0).
 Tool T0 is not allowed.

Warning: Pre-position the tool so that there can be no collision with the workpiece or clamping devices.

9.2 Description of addresses

Mandatory addresses

Mandatory addresses are shown in black.
 If a mandatory address is not entered an error message is issued.

Optional addresses

Optional addresses are shown in light grey.
 If this address is not entered it is ignored or given the basic setting that has already been entered.

Explanation of addresses.

The addresses described here are used in most cycles. Specific addresses are described in the cycle.

X, Y, Z: Starting point

Starting point of measuring motion. The measuring cycle starts here. If all the starting point coordinates are not entered, the current position of the tool is adopted.

Execution

Unlike a milling cycle, a measuring cycle is carried out directly from the starting point (X, Y, Z). The probe moves to the first starting point (X, Y, Z) in rapid motion and depending on G28, using positioning logic.

C1= Maximum measured length

Maximum distance between the starting and finishing points of the measuring stroke. (Basic setting 10). Movement stops once the wall of the workpiece or the end of the measured length is reached.

Note:

If there is no contact with material within the measuring stroke (C1=) an error message is issued.

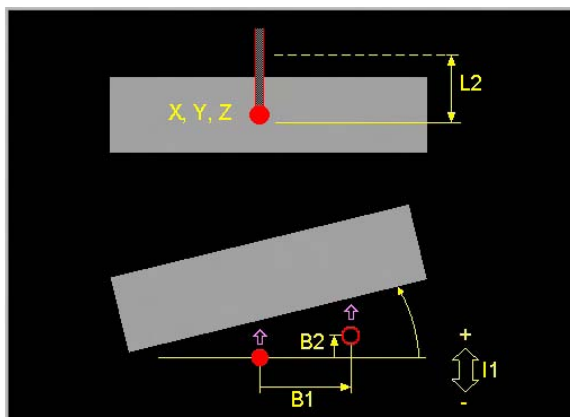
L2= Safety distance

During (if I3=1) and at the end of measurement, the probe moves at the safety distance (default setting 0 for measurement on the outside of the workpiece or 1mm for measurements in pockets and holes). Safety distance (L2=) is with respect to the current starting point X, Y, Z.

- B3=** Distance to the corner
 The distance between the first starting point and the corner of the workpiece.
- Distance to next measurement about the corner of the workpiece.
 The path traced by the probe around the corner of the workpiece to the starting point of the 2nd measurement is the same length in both directions. For each direction the distance is the sum of B3= and the first measuring distance travelled.
- I1=** Direction of probe movement with respect to workpiece
 I1=±1 Main axis
 I1=±2 Secondary axis
 I1=-3 Tool axis
 The angular reference axes are always perpendicular to the direction of scanning
- I3=** Movement between measuring strokes.
 I3= is used to determine whether the positioning movements between measurements take place at the measuring height or the safety distance (L2=).
- I3=0 The positioning movement between measuring strokes is at the measuring height and parallel to the main axis.
 In the case of circular movement the positioning movement is circular and at the feed rate.
- I3=1 The positioning movement between measuring strokes is at the measuring height and in a line between measurement points.
- I4=** Corner number (1 - 4)
 Defines the corner where the first measurement should take place (default setting 1).
 The first measurement is always perpendicular to the main axis.
 The second measurement is always perpendicular to the secondary axis.
- O1= to O6=** Save measured values
 The measured values can be written in the E parameters.. The number of the E parameter must be entered. If no number is entered, nothing is saved.
 Example: O1=10 means that the result is stored in E parameter 10.
- F2=** Measuring feed The basic setting is MC843.

9.3 G620 Angle measurement

Measuring the inclined position of a clamped workpiece.



```
G   Angle measurement
I1= Meas.dir. ±1/±2/-3=main/minor/T1
X   Starting point
Y   Starting point
Z   Starting point
B1= Dist. meas. positions main axis
B2= Dist. meas. positions par. axis
C1= Measuring distance
L2= Safety distance
I3= 2nd measur. via L2 0=no 1=yes
I5= G5x offset 0=no 1=B4 2=A/B/C
O3= E-Par. measured angle
F2= Measuring feed
A1= Target value angle
```

- B1= Distance with direction along the main axis.
If I1=±2, B1= must be programmed (B1= must not equal zero).
If I1=-3, B1= and B2= do not both need to be programmed at the same time.
- B2= Distance with direction along the secondary axis.
If I1=±1, B2= must be programmed (B2= must not equal zero).
If I1=-3, B1= and B2= do not both need to be programmed at the same time.
The following is not allowed: B1= B2= 0
- I5= Save measured values in a zero point offset.
I5=0 Do not save
I5=1 Save in the active zero point offset in the angle of rotation (G54 B4=).
I5=2 Save in the active zero point offset in the axis of rotation (A/B/C).
On saving, the measured values are added to the active zero point offset.
- A1= If the measured angle is saved in the active zero point offset (I5>0), it is used to calculate the target value.
The measured position thus becomes the target value for subsequent programming.
The other addresses are described in the introduction to the measuring cycles.

Basic settings

B1=0, B2=0, C1=10, L2=0, I3=0, I5=0, F2=MC843, A1=0.

Notes and application

Depending on the plane selected (G17, G18 or G19), the parameter I1= determines the direction of measurement and this defines the meanings of B1= and B2=.

	G17				G18				G19			
Direction of measurement	I1=±1	I1=±2	I1=3		I1=±1	I1=±2	I1=3		I1=±1	I1=±2	I1=3	
			B1=	B2=			B1=	B2=			B1=	B2=
Angle plane	XY	XY	XZ	YZ	XZ	XZ	XY	ZY	YZ	YZ	YX	ZX
Axis of rotation	C	C	B	A	B	B	C	A	A	A	C	B

EASYoperate ⇔ DIN/ISO

The addresses O3= and F2= are not available in EASYoperate.

The cycle

1. Rapid motion to the first starting point (X, Y, Z). If X, Y or Z is not programmed, the current position is taken as the starting point.
2. First measurement with measuring feed (F2=) until the end of the workpiece or the maximum measuring distance (C1=) is reached.
3. Rapid movement back to the starting point. An error message is issued if the probe has not switched within the maximum measuring distance (C1=).
4. Rapid motion, depending on I3= over the safety distance (L2=) to the starting point for the 2nd measurement.
5. Second measurement (as points 2 and 3).
6. At the end there is rapid movement to the safety distance (L2=).
7. The measured value is stored as per I5=.

Example: Setting up a workpiece

N40 G17

Set the surface plane

N50 G54 I3

Set zero

N60 G620 X-50 Y-50 Z-5 I1=2

B1=100 L2=10 I3=1 I5=2

Define and execute the measuring cycle

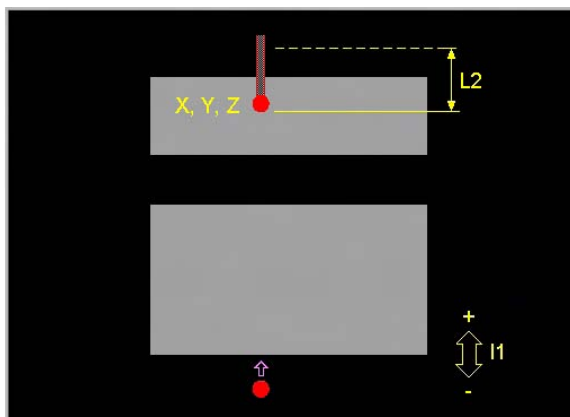
After the cycle G54 I3 is recalculated

N70 G0 C0

Rotary table is positioned at zero (G17).

9.4 G621 Position measurement

Measurement of a coordinate on the wall of a workpiece.



```
G    Position measurement
I1=  Meas.dir. ±1/±2/-3=main/minor/T1
X    Starting point
Y    Starting point
Z    Starting point
C1=  Measuring distance
L2=  Safety distance
I5=  G5x offset 0=no 1=X/Y/Z
O1=  E-Par. for measured position
F2=  Measuring feed
B1=  Target position
```

I5= Save measured values in a zero point offset.

I5=0 Do not save

I5=1 Save in the active zero point offset in the linear axes (X/Y/Z).

On saving, the measured values are added to the active zero point offset.

B1= If the measured coordinate is saved in the active zero point offset (I5>0), it is used to calculate the target value.

The measured coordinate thus becomes the target value for subsequent programming.

The other addresses are described in the introduction to the measuring cycles.

Basic settings

C1=10, L2=0, I5=0, F2=MC843, B1=0

Notes and application

Address I1= determines the direction of measurement, depending on the plane selected (G17, G18 or G19).

EASYoperate ⇔ DIN/ISO

The addresses O1= and F2= are not available in EASYoperate.

The cycle

- 1 Rapid motion to the first starting point (X, Y, Z). If X, Y or Z is not programmed, the current position is taken as the starting point.
- 2 First measurement with measuring feed (F2=) until the end of the workpiece or the maximum measuring distance (C1=) is reached.
- 3 Rapid movement back to the starting point. An error message is issued if the probe has not switched within the maximum measuring distance (C1=).
- 4 At the end, rapid movement back to the safety distance (L2=).
- 5 The measured value is stored as per I5=.

Example: Measuring a position.

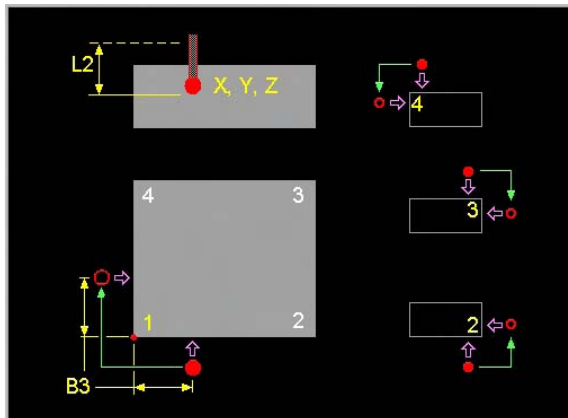
```
N60 G621 X40 Y40 Z-5 I1=2
      L2=20 O1=300
```

Define and execute the measuring cycle

After the cycle the result is written in E parameter (E300).

9.5 G622 Corner outside measurement

Measure the corner position (outside) of an aligned workpiece.



G Corner outside measurement
 I4= Corner number
 X Starting point
 Y Starting point
 Z Starting point
 B3= Distance to corner
 C1= Measuring distance
 L2= Safety distance
 I3= 2nd measurem. via L2 0=no 1=yes
 I5= G5x offset 0=no 1=X/Y/Z
 O1= E-Par. meas. position main axis
 O2= E-Par. meas. position minor axis
 F2= Measuring feed
 X1= Target position corner
 Y1= Target position corner

Z1= Target position corner

I5= Save measured values in a zero point offset

I5=0 Do not save

I5=1 Save in the active zero point offset in the linear axes (X/Y/Z).

On saving, the measured values are added to the active zero point offset.

X1=, Y1=, Z1= If the measured coordinate is saved in the active zero point offset (I5>0), it is used to calculate the target value.

The measured coordinate thus becomes the target value for subsequent programming.

The other addresses are described in the introduction to the measuring cycles.

Basic settings

I4=1, B3=10, C1=10, L2=0, I3=0, I5=0, F2=MC843, X1=0, Y1=0, Z1=0.

Notes and application

Check:

- the sides must be parallel to the axes
- the angle of the workpiece must be 90 degrees
- the measured plane is at right angles to the axis of the workpiece.

Direction of approach to measurements

- the first measurement is always perpendicular to the main axis.
- the second measurement is always perpendicular to the secondary axis.

Remark: The support picture is in G17. By a machine with exchanged axis (G18) the picture is not correct. The angle 1 will be exchanged with 2 and 3 with 4.

EASYoperate ⇔ DIN/ISO

The addresses O1=, O2= and F2= are not available in EASYoperate.

The cycle

- 1 Rapid motion to the first starting point (X, Y, Z). If X, Y or Z is not programmed, the current position is taken as the starting point.
- 2 First measurement with measuring feed (F2=) until the end of the workpiece or the maximum measuring distance (C1=) is reached.
- 3 Rapid movement back to the first starting point. An error message is issued if the probe has not switched within the maximum measuring distance (C1=).

- 4 Rapid motion, depending on I3= over the safety distance (L2=) to the starting point for the 2nd measurement.
- 5 Second measurement (as points 2 and 3).
- 6 At the end, rapid movement back to the safety distance (L2=).
- 7 The measured value is stored as per I5=.

Example: Setting up an outside corner of a workpiece

N40 G1 X.. Y.. Z-5

Locate the probe 10mm to the right of corner 1 and 8mm away from the front.

N50 G54 I3

Set zero

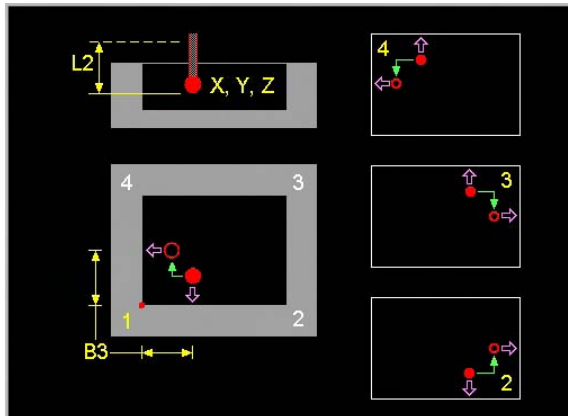
N60 G622 L2=20 B3=25 I3=1
I5=1 X1=-50 Y1=-50

Define and execute the measuring cycle

After the measuring cycle the zero point offset is overwritten so that the coordinates of corner 1 are equal to X1= and Y1=.

9.6 G623 Corner inside measurement

Measure the corner position (inside) of an aligned workpiece.



```
G   Corner inside measurement
I4= Corner number
X   Starting point
Y   Starting point
Z   Starting point
B3= Distance to corner
C1= Measuring distance
L2= Safety distance
I3= 2nd measurem. via L2 0=no 1=yes
I5= G5x offset 0=no 1=X/Y/Z
O1= E-Par. meas. position main axis
O2= E-Par. meas. position minor axis
F2= Measuring feed
X1= Target position corner
Y1= Target position corner
```

```
Z1= Target position corner
```

I5= Save measured values in a zero point offset

I5=0 Do not save

I5=1 Save in the active zero point offset in the linear axes (X/Y/Z).

On saving, the measured values are added to the active zero point offset.

X1=, Y1=, Z1= If the measured coordinate is saved in the active zero point offset (I5>0), it is used to calculate the target value.

The measured coordinate thus becomes the target value for subsequent programming.

The other addresses are described in the introduction to the measuring cycles.

Basic settings

I4=1, B3=10, C1=10, L2=10, I3=0, I5=0, F2=MC843, X1=0, Y1=0, Z1=0.

Notes and application

Check:

- the sides must be parallel to the axes
- the workpiece angle must be 90 degrees
- the measured plane is at right angles to the axis of the workpiece.

Direction of approach to measurements

- the first measurement is always perpendicular to the main axis.
- the second measurement is always perpendicular to the secondary axis.

Remark: The support picture is in G17. By a machine with exchanged axis (G18) the picture is not correct. The angle 1 will be exchanged with 2 and 3 with 4.

EASYoperate ⇔ DIN/ISO

The addresses O1=, O2= and F2= are not available in EASYoperate.

The cycle

1. Rapid motion to the first starting point (X, Y, Z). If X, Y or Z is not programmed, the current position is taken as the starting point.
2. First measurement with measuring feed (F2=) until the end of the workpiece or the maximum measuring distance (C1=) is reached.
3. Rapid movement back to the first starting point. An error message is issued if the probe has not switched within the maximum measuring distance (C1=).

4. Rapid motion, depending on I3= over the safety distance (L2=) to the starting point for the 2nd measurement.
5. Second measurement (as points 2 and 3).
6. At the end, rapid movement back to the safety distance (L2=).
7. The measured value is stored as per I5=.

Example: Setting up an inside corner of a workpiece

N40 G1 X.. Y.. Z-5

Locate the probe 10mm to the right of corner 1 and 8mm away from the front.

N50 G54 I3

Set zero.

N60 G623 L2=20 B3=25 I3=1

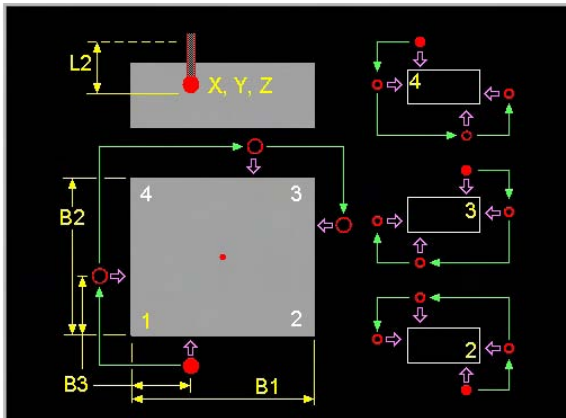
I5= 1 X1=-50 Y1=-50

Define and execute the measuring cycle

After the measuring cycle the zero point offset is overwritten so that the coordinates of corner 1 are equal to X1= and Y1=.

9.7 G626 Datum outside rectangle

Measuring the centre of an axially parallel rectangle.



G Datum outside rectangle
 I4= Corner number
 X Starting point
 Y Starting point
 Z Starting point
 B1= 1st Side length
 B2= 2nd Side length
 B3= Distance to corner
 C1= Measuring distance
 L2= Safety distance
 I3= 2nd measurem. via L2 0=no 1=yes
 I5= G5x offset 0=no 1=X/Y/Z
 O1= E-Par. meas. centre main axis
 O2= E-Par. meas. centre minor axis
 O4= E-Par. meas. length main axis
 O5= E-Par. meas. length minor axis
 F2= Measuring feed
 X1= Target centre point
 Y1= Target centre point
 Z1= Target centre point

I5= Save measured values in a zero point offset

I5=0 Do not save

I5=1 Save in the active zero point offset in the linear axes (X/Y/Z).

On saving, the measured values are added to the active zero point offset.

X1=, Y1=, Z1= If the measured coordinate is saved in the active zero point offset (I5>0), it is used to calculate the target value.

The measured coordinate thus becomes the target value for subsequent programming.

The other addresses are described in the introduction to the measuring cycles.

Basic settings

I4=1, B3=10, C1=10, L2=0, I3=0, I5=0, F2=MC843, X1=0, Y1=0, Z1=0.

Notes and application

Two opposite corners of the workpiece are measured (1+3 or 2+4)

Direction of approach to the first corner measurement

- the first measurement is always perpendicular to the main axis.

- the second measurement is always perpendicular to the secondary axis

Direction of approach to the second corner measurement

- clockwise from corner number 1 → 3 or 3 → 1

- anticlockwise from corner number 2 → 4 or 4 → 2

Remark: The support picture is in G17. By a machine with exchanged axis (G18) the picture is not correct. The angle 1 will be exchanged with 2 and 3 with 4.

EASYoperate ⇔ DIN/ISO

The addresses O1=, O2=, O4=, O5= and F2= are not available in EASYoperate.

The cycle

1. Rapid motion to the first starting point (X, Y, Z). If X, Y or Z is not programmed, the current position is taken as the starting point.

2. First measurement with measuring feed (F2=) until the end of the workpiece or the maximum measuring distance (C1=) is reached.
3. Rapid movement back to the starting point. An error message is issued if the probe has not switched within the maximum measuring distance (C1=).
4. Rapid motion, depending on I3= over the safety distance (L2=) to the starting point for the 2nd measurement.
5. Second measurement (as points 2 and 3).
6. The opposite corner is measured using 3rd and 4th measurements (as points 2 and 3).
7. At the end, rapid movement back to the safety distance (L2=).
8. The measured value is stored as per I5=.

Example: Save the centre of a rectangle in the zero point offset.

N50 G54 I3

Set zero

N60 G626 X-45 Y-3 Z-5 B1=100

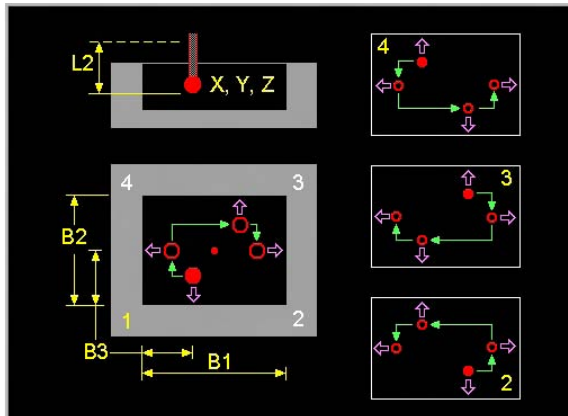
B2=20 B3=5 I3=1 I5=1

Define and execute the measuring cycle

After the cycle X and Y are recalculated in G54 I3

9.8 G627 Datum inside rectangle

Measuring the centre of an axially parallel rectangular hole.



G Datum inside rectangle
 I4= Corner number
 X Starting point
 Y Starting point
 Z Starting point
 B1= 1st Side length
 B2= 2nd Side length
 B3= Distance to corner
 C1= Measuring distance
 L2= Safety distance
 I3= 2nd measurem. via L2 0=no 1=yes
 I5= G5x offset 0=no 1=X/Y/Z
 O1= E-Par. meas. centre main axis
 O2= E-Par. meas. centre minor axis
 O4= E-Par. meas. length main axis
 O5= E-Par. meas. length minor axis
 F2= Measuring feed
 X1= Target centre point
 Y1= Target centre point
 Z1= Target centre point

I5= Save measured values in a zero point offset

I5=0 Do not save

I5=1 Save in the active zero point offset in the linear axes (X/Y/Z).

On saving, the measured values are added to the active zero point offset.

X1=, Y1=, Z1= If the measured coordinate is saved in the active zero point offset (I5>0), it is used to calculate the target value.

The measured coordinate thus becomes the target value for subsequent programming.

The other addresses are described in the introduction to the measuring cycles.

Basic settings

I4=1, B3=10, C1=10, L2=10, I3=0, I5=0, F2=MC843, X1=0, Y1=0, Z1=0.

Notes and application

Two opposite corners of the workpiece are measured (1+3 or 2+4)

Direction of approach to the first corner measurement

- the first measurement is always perpendicular to the main axis.

- the second measurement is always perpendicular to the secondary axis.

Direction of approach to the second corner measurement

- clockwise from corner number 1 → 3 or 3 → 1

- anticlockwise from corner number 2 → 4 or 4 → 2

Remark: The support picture is in G17. By a machine with exchanged axis (G18) the picture is not correct. The angle 1 will be exchanged with 2 and 3 with 4.

EASYoperate ⇔ DIN/ISO

The addresses O1=, O2=, O4=, O5= and F2= are not available in EASYoperate.

The cycle

1. Rapid motion to the first starting point (X, Y, Z). If X, Y or Z is not programmed, the current position is taken as the starting point.

2. First measurement with measuring feed (F2=) until the end of the workpiece or the maximum measuring distance (C1=) is reached.
3. Rapid movement back to the starting point. An error message is issued if the probe has not switched within the maximum measuring distance (C1=).
4. Rapid motion, depending on I3= over the safety distance (L2=) to the starting point for the 2nd measurement.
5. Second measurement (as points 2 and 3).
6. The opposite corner is measured using 3rd and 4th measurements (as points 2 and 3).
7. At the end, rapid movement back to the safety distance (L2=).
8. The measured value is stored as per I5=.

Example: Save the centre of a rectangle in the zero point offset.

N50 G54 I3

Set zero

N60 G627 X-45 Y-3 Z-5 B1=100

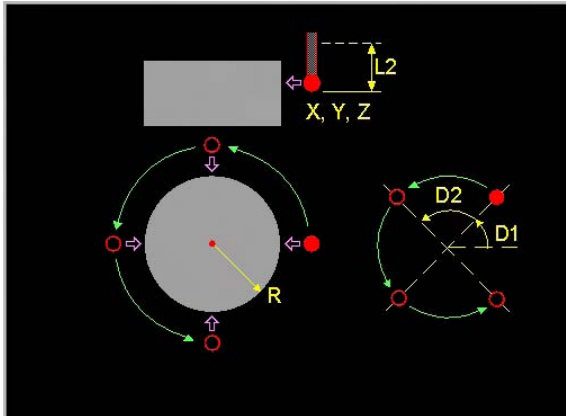
B2=20 B3=5 I3=1 I5=1

Define and execute the measuring cycle

After the cycle X and Y are recalculated in G54 I3

9.9 G628 Circle measurement outside

Measuring the centre of a circle.



G Circle measurement outside
 R Circle radius
 X Starting point
 Y Starting point
 Z Starting point
 D1= Starting angle
 D2= Second angle
 C1= Measuring distance
 L2= Safety distance
 I2= Probe orientat. 0=no 1=180 2=yes
 I3= 2nd measur. via L2 0=no 1=yes
 I5= G5x offset 0=no 1=X/Y/Z
 O1= E-Par. meas. centre main axis
 O2= E-Par. meas. centre minor axis
 O6= E-Par. measured diameter

F2= Measuring feed
 X1= Target centre point
 Y1= Target centre point
 Z1= Target centre point

D1= Angular offset of the circle measurement with respect to the main axis.

I2= Probe orientation in the direction of measurement:

0= measurement without rotation

1= measurement using 2 measurements with 180° rotation.

First measurement with standard orientation (MC849).

Second measurement with 180° rotation

The measured value is the average of these two.

2= measurement with orientation in the direction of measurement.

Only possible with an infra-red probe with all-round emitter.

The orientation option for the probe is defined in MC486.

I5= Save measured values in the zero point offset

0 Do not save

1 Save in the active zero point offset in the linear axes (X/Y/Z).

On saving, the measured values are added to the active zero point offset.

X1=, Y1=, Z1= If the measured coordinate is saved in the active zero point offset (I5>0), it is used to calculate the target value.

The measured coordinate thus becomes the target value for subsequent programming.

The other addresses are described in the introduction to the measuring cycles.

Basic settings

D1=0, D2=90, C1=20, L2=10, I2=0, I3=0, I5=0, F2=MC843, X1=0, Y1=0, Z1=0.

Notes and application

The starting point selected for circle measurement should be such that the first measurement moves as exactly as possible in the direction of the centre of the circle.

Circle measurement is executed anticlockwise.

EASYoperate ⇔ DIN/ISO

The addresses O1=, O2=, O6= and F2= are not available in EASYoperate.

The cycle

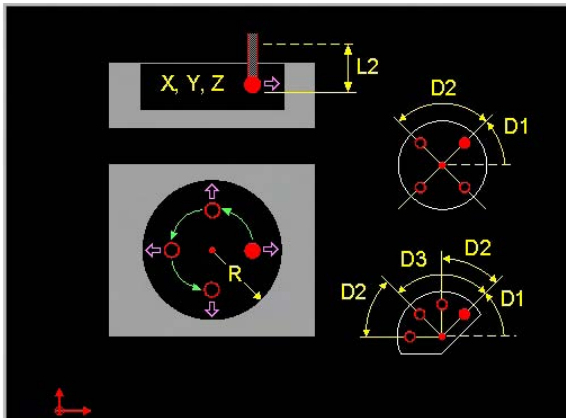
1. Rapid motion to the first starting point (X, Y, Z). If X, Y or Z is not programmed, the current position is taken as the starting point.
2. First measurement with measuring feed (F2=) until the end of the workpiece or the maximum measuring distance (C1=) is reached.
3. Rapid movement back to the starting point. An error message is issued if the probe has not switched within the maximum measuring distance (C1=).
4. Rapid motion, depending on I3= over the safety distance (L2=) to the starting point for the 2nd measurement.
5. Second, 3rd and 4th measurements (as points 2 to 4).
6. At the end, rapid movement back to the safety distance (L2=).
7. The measured value is stored as per I5=.

Example: Save the centre of a circular projection in the zero point offset.

N50 G54 I3	Set zero
N60 G628 X-45 Y-3 Z-5 R50 I3=1 I5=1	Define and execute the measuring cycle
	After the cycle X and Y are recalculated in G54 I3

9.10 G629 Circle measurement inside

Measuring the centre of a circular hole.



```
G Circle measurement inside
R Circle radius
X Starting point
Y Starting point
Z Starting point
D1= Starting angle
D2= Second angle
D3= Third angle
C1= Measuring distance
L2= Safety distance
I2= Probe orientat. 0=no 1=180 2=yes
I3= 2nd measurem. via L2 0=no 1=yes
I5= G5x offset 0=no 1=X/Y/Z
O1= E-Par. meas. centre main axis
O2= E-Par. meas. centre minor axis
```

```
O6= E-Par. measured diameter
F2= Measuring feed
X1= Target centre point
Y1= Target centre point
Z1= Target centre point
```

- D1= Angular offset of the circle measurement with respect to the main axis.
 D2= Angle between the first and the second measurement and between the third and fourth measurement. The lowest value is 5°.
 D3= Angle between the first and the third measurement. D3 must be at least 5° bigger than D2. When D3 and D2 are equal, a 3-points measurement is executed.

Remark: The highest accuracy will be reached by a symmetrical measuring with default values D2=90 and D3=180.

- I2= Probe orientation in the direction of measurement:
 0= Measurement without rotation
 1= measurement using 2 measurements with 180° rotation.
 First measurement with standard orientation (MC849).
 Second measurement with 180° rotation
 The measured value is the average of these two.
 2= measurement with orientation in the direction of measurement.
 Only possible with an infra-red probe with all-round emitter.
 The orientation option for the probe is defined in MC486.
- I5= Save measured values in the zero point offset
 I5=0 Do not save
 I5=1 Save in the active zero point offset in the linear axes (X/Y/Z).
 On saving, the measured values are added to the active zero point offset.
- X1=, Y1=, Z1= If the measured coordinate is saved in the active zero point offset (I5>0), it is used to calculate the target value.
 The measured coordinate thus becomes the target value for subsequent programming.
 The other addresses are described in the introduction to the measuring cycles.

Basic settings

D1=90, D2=90, D3=180, C1=10, L2=10, I2=0, I3=0, I5=0, F2=MC843, X1=0, Y1=0, Z1=0.

Notes and application

The starting point selected for circle measurement should be such that the first measurement moves as exactly as possible in the direction of the centre of the circle.

Circle measurement is executed anticlockwise.

EASYoperate ⇔ DIN/ISO

The addresses O1=, O2=, O6= and F2= are not available in EASYoperate.

The cycle

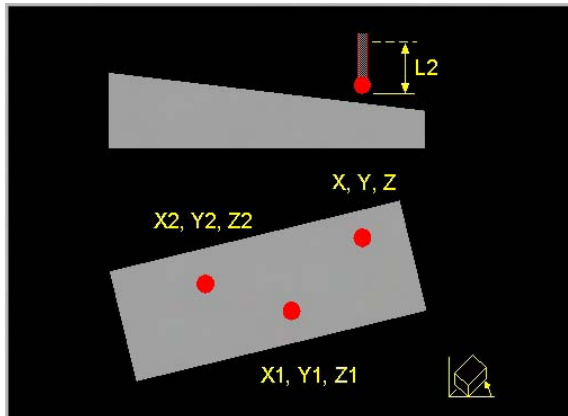
1. Rapid motion to the first starting point (X, Y, Z). If X, Y or Z is not programmed, the current position is taken as the starting point.
2. First measurement with measuring feed (F2=) until the end of the workpiece or the maximum measuring distance (C1=) is reached.
3. Rapid movement back to the starting point. An error message is issued if the probe has not switched within the maximum measuring distance (C1=).
4. Rapid motion, depending on I3= over the safety distance (L2=) to the starting point for the 2nd measurement.
5. Third and 4th measurements (as points 2 to 4).
6. At the end, rapid movement back to the safety distance (L2=).
7. The measured value is stored as per I5=.

Example: Save the centre of a circle in the zero point offset.

N50 G54 I3	Set zero
N60 G629 X-45 Y-3 Z-5 R50 I3=1 I5=1	Define and execute the measuring cycle
	After the cycle X and Y are recalculated in G54 I3

9.11 G631 Measure position of inclined plane

Measure the inclination of a workpiece plane surface (g7) using 3-point measurement.



```
G      Obliqueness measurement
I1=    Meas.dir. ±1/±2/-3=main/minor/T1
X      Starting point (meas. point 1)
Y      Starting point (meas. point 1)
Z      Starting point (meas. point 1)
X1=    Measuring point 2
Y1=    Measuring point 2
Z1=    Measuring point 2
X2=    Measuring point 3
Y2=    Measuring point 3
Z2=    Measuring point 3
O1=    E-Par. Angle of abs. rotation A5=
O2=    E-Par. Angle of abs. rotation B5=
O3=    E-Par. Angle of abs. rotation C5=
C1=    Measuring distance
```

```
L2=    Safety distance
I3=    Measur. 2 and 3 via L2 0=no 1=yes
F2=    Measuring feed
```

L2= The safety measurement is related to each starting point of a measurement and is in the measuring direction.

The other addresses are described in the introduction to the measuring cycles.

Basic settings

C1=20, L2=0, I3=0, F2=MC843

Notes and application

The measured inclination can be set exactly with the G7 function.

EASYoperate ⇔ DIN/ISO

The addresses O1=, O2=, O3= and F2= are not available in EASYoperate.

The cycle

Rapid movements always take place with positioning logic in the active (and possible already tilted) machining plane.

1. Rapid motion to the first starting point (X, Y, Z).
2. First measurement with measuring feed (F2=) until the end of the workpiece or the maximum measuring distance (C1=) is reached.
3. Rapid movement back to the starting point. An error message is issued if the probe has not switched within the maximum measuring distance (C1=).
4. Movement, depending on I3=, over the safety distance (L2=) to the starting point for the 2nd measurement.
5. Second and 3rd measurements (as points 2 to 4).
6. At the end there is rapid movement to the safety distance (L2=).
7. The measured values are stored.

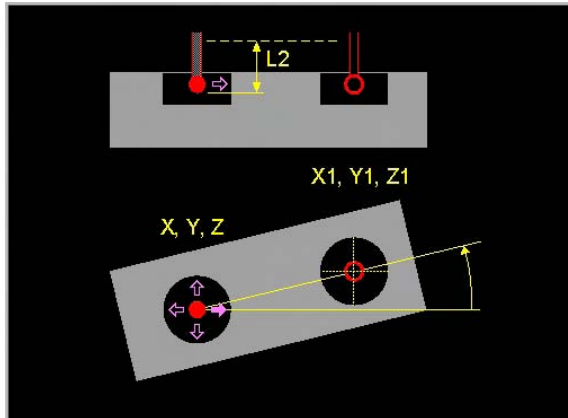
Example: Set up the machining plane and rotate

N3416	Measure the machining plane and rotate
N1 G17	Set the surface plane
N2 G54 I1	
N3 T35 M66	Change the probe
N4 G0 X50 Y20 Z100	
N5 G631 X18 Y0 Z-16 X1=18 Y1=10	
Z1=-16 X2=10 Y2=0 Z2=-6 C1=15	
L2=20 O1=10 O2=11 O3=12 F2=150	Measure position of inclined plane
N10 G0 Z100	Go to a safe height (G17)
N11 G7 A5=E10 B5=E11 C5=E12 L1=1	Turn the machining plane

9.12 G633 Angle measurement 2 holes

Measuring the skew of a work piece set-up.

The probe measures the centre points of two cylindrical holes. Next the MillPlus calculates the angle between the main axis of the working plane and the connection line between the centre points of the holes.



```
G   Angle measurement 2 holes
X   Starting point (meas. point 1)
Y   Starting point (meas. point 1)
Z   Starting point (meas. point 1)
X1= Measuring point 2
Y1= Measuring point 2
Z1= Measuring point 2
L2= Safety distance
C1= Measuring distance
I5= G5x offset 0=no 1=B4 2=A/B/C
O3= E-Par. measured angle
F2= Measuring feed
A1= Target value angle
```

X, Y, Z Starting point of the measurement of the first cylindrical hole (or the actual position)

X1=, Y1=, Z1= Starting point of the measurement of the second cylindrical hole (all three coordinates must be entered)

C1= Maximum measuring distance

L2= Safety distance

O3= Number of the E-parameter in which the angle is stored.

I5= Storing the measuring values in a zero point shift:

I5=0 Do not store

I5=1 Store in the active zero point shift of the rotation angle (B4=).

I5=2 Store in the active zero point shift of the rotary axis (A/B/C).

During storing the measuring values are added to the active zero point shift.

A1= If the measured angle is stored in the active zero point shift (I5>0), it is calculated in the command position.

For the remaining programming the measured position gets the command position.

The description of the remaining addresses can be found in the introduction to measuring cycles.

Default settings

C1=20, I5=0, F2=MC_0843, A1=0.

Notes and usage

The starting position must be programmed inside the cylindrical hole.

EASYoperate ⇔ DIN/ISO

In EASYoperate the addresses O3= and F2= are not available.

Cycle sequence

1. Movement in rapid to the first starting point (X, Y, Z) in the first cylindrical hole. When X, Y, Z are not programmed, the actual position is taken as the starting point.
2. Measuring movement with measuring feed (F2=) to the hole side or till the maximum measuring distance (C1=) is reached. The centre point is first measured roughly and then exactly
3. Movement in rapid back to the starting position. An error message is given when the measuring probe was not triggered within the maximum measuring distance (C1=). Retract movement to the safety distance (L2=)
4. Movement in rapid with regard to the safety distance to the starting point of the 2nd hole.
5. The second hole is measured in the same way.

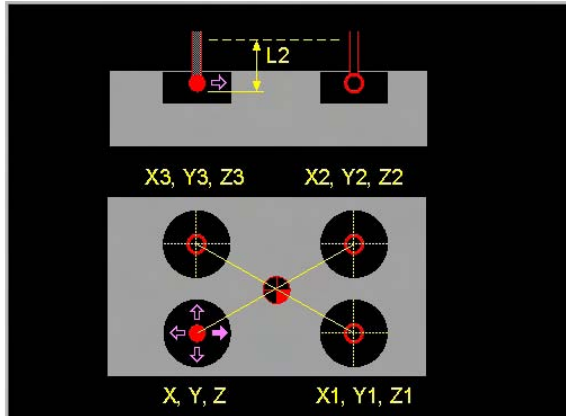
6. At the end a movement in rapid follows to the safety distance (L2=).
7. Depending on I5= the measured value is stored.

Example: Aligning a work piece

N40 G17	Set the plane
N50 G54 I3	Set the zero point
N60 G633 X-100 Y-50 Z-5	Define the measuring cycle with the starting point of the 1st cylindrical hole
X1=-10 Y1=-50 Z1=-5	Starting point of the 2nd hole
L2=30 I5=2	Safety distance = 30 and the measured value is stored in the zero point shift of the rotary table (C)
N70 G0 C0	Rotary table is positioned to zero (G17)

9.13 G634 Measurement center 4 holes

This measurement cycle calculates the intersection point of the connection lines of two cylindrical hole center points and sets this interconnection point as a centre point. At choice the MillPlus can store the interconnection point also in a zero point table.



```
G   Measurement center 4 holes
X   Starting point (meas. point 1)
Y   Starting point (meas. point 1)
Z   Starting point (meas. point 1)
X1=  Measuring point 2
Y1=  Measuring point 2
Z1=  Measuring point 2
X2=  Measuring point 3
Y2=  Measuring point 3
Z2=  Measuring point 3
X3=  Measuring point 4
Y3=  Measuring point 4
Z3=  Measuring point 4
L2=  Safety distance
C1=  Measuring distance
```

```
I5=  G5x offset 0=no 1=X/Y/Z
O1=  E-Par. meas. centre main axis
O2=  E-Par. meas. centre minor axis
F2=  Measuring feed
X4=  Target centre point
Y4=  Target centre point
Z4=  Target centre point
```

- X, Y, Z Starting point of the measurement of the 1st hole (or the actual position)
 X1=, Y1=, Z1= Starting point of the measurement of the 2nd hole (all 3 coordinates must be entered)
 X2=, Y2=, Z2= Starting point of the measurement of the 3rd hole (all 3 coordinates must be entered)
 X3=, Y3=, Z3= Starting point of the measurement of the 4th hole (all 3 coordinates must be entered)
 C1= Maximum measuring distance
 L2= Safety distance
 I5= Storing measuring values in a zero point shift:
 I5=0 Do not store
 I5=1 Store in the active zero point shift of the linear axes (X/Y/Z).
 During storing the measuring values are added to the active zero point shift.
 X4=, Y4=, Z4= If the measured coordinate is saved in the active zero point offset (I5>0), it is used
 to calculate the target value.
 The measured coordinate thus becomes the target value for subsequent programming.
 O1= Number of the E-parameter in which the measured centre point in the main axis is stored.
 O2= Number of the E-parameter in which the measured centre point of the minor axis is stored.

The description of the remaining addresses can be found in the introduction to measuring cycles.

Default settings

C1=20, I5=0, F2=MC_0843.

Notes and usage

The starting position must be programmed inside the cylindrical hole.

EASYoperate ⇔ DIN/ISO

In EASYoperate the addresses O1=, O2= and F2= are not available.

Cycle sequence

1. Movement with rapid to the first starting point (X, Y, Z) in the 1st cylindrical hole. When X, Y, Z are not programmed the actual position is taken as starting point.
2. Measuring movement with measuring feed (F2=) to the hole side or till the maximum measuring distance (C1=) is reached. The centre point is first measured roughly and then exactly.
3. Movement in rapid back to the starting position. An error message is given when the measuring probe was not triggered within the maximum measuring distance (C1=). Retract movement to the safety distance (L2=)
4. Movement in rapid with regard to the safety distance to the starting point of the 2nd hole.
5. The second hole is measured in the same way.
6. To measure the 3rd and 4th hole the steps 3 and 4 are repeated.
8. At the end a movement in rapid follows to the safety distance (L2=).
9. Depending on I5= the measured value is stored.

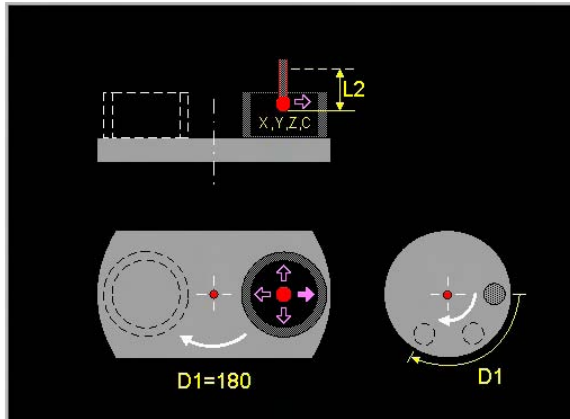
Example: Determine the centre point of 4 cylindrical holes of a work piece

N40 G17	Set the plane
N50 G54 I3	Set the zero point
N60 G634	Define the measuring cycle with
X-10 Y-20 Z-5	Starting point of the 1st hole
X1=-100 Y1=-40 Z1=-5	Starting point of the 2nd hole
X2=-100 Y2=-100 Z2=-5	Starting point of the 3rd hole
X3=-10 Y3=-120 Z3=-5	Starting point of the 4th hole
L2=30 I5=1	Safety distance is 30. After the measuring cycle X and Y in G54 are updated.

9.14 G640 Locate table rotation center.

Measuring and correction of temperature dependant (or small mechanical) table displacements with the help of a measuring probe. (TPC= Table Position Control)

For this measurement a hole in the table or work piece must be present. The probe measures the hole, the table is rotated 180 degrees and the measurement is repeated. The cycle G640 corrects the, from the measurement calculated turning center in both axes.



```
G  Locate table rotation center
C1= Max. measuring distance
X  Starting point
Y  Starting point
Z  Starting point
C  C-coordinate rotary table
D1= End angle
L2= Safety distance
I1= Kin. elements 0=clear 1=measure
I2= Suppress correction 0=no 1=yes
O1= E-Parameter offset X axis
O2= E-Parameter offset Y axis
```

D1 End angle.

This end angle is necessary by C-axis with limited reach (Z.B. set up table).

When D1 between -180 and +180, the measuring will be done on 3 positions.

When D1 equal -180 or +180 is, the measuring will be done on 2 positions.

When the measuring happens on 3 positions, which are not lying on a circle, but on an arc, the calculation of the table rotation centre is not so precise as with 2 opposite holes.

Basic settings

I1=1, I2=0, L2=0, D1=180

Notes and application

Remarks

- C Axis must be present.
- The starting position must be programmed inside the hole.
- The deviation measured in the X and Y axis, is corrected in the first correction element of the relevant axis in the active kinematics model.
- When G7 is active, X, Y, Z und C must be entered.

It is not allowed to program G640 when:

- G18, G19, G36, G182 are active.
- G54 up to G59 B4= does not equal 0.
- G93 B4= is programmed with A or B or C.
- Tool number T0 is programmed.

G640 activates: G90, G40, G39 L0 R0, G72

G640 deactivates: G7

All measurement movements are performed with the default measuring feed (MC842).

Conditions

- The kinematics model of the machine tool must be entered and must contain the correction elements for X and Y.
- The maximum correction per axis is $\pm 0.200\text{mm}$.

Switching on:

The correction elements of the kinematics model are set to zero when switching on the machine tool.

Cycle sequence

- 1 When G7 is active or the rotary axes are not at the zero point position:
 - Retract movement with rapid to the SW-end switch
 - G7 is switched off
 - B axis and A axis are moved to the zero point position and the tool axis is moved again to the SW end switch
 In all other cases:
 - Retract movement with rapid to the SW end switch or when programmed to the safety distance (L2=). If the measuring probe is already in the start position (X,Y,Z and C not programmed), this movement is skipped.
- 2 Movement with rapid to the start position in the hole. Measurement of the center point.
- 3 Second measurement to measure the center point exactly (sequence depends on the probe type).
- 4 Retract movement with rapid to the SW end switch or when programmed to the safety distance (L2=). When the hole in the turning center is used, no retract movements occur.
- 5 The rotary table is rotated over 180°.
- 6 The hole is measured in the new position in the same way.
- 7 Retract movement with rapid to the SW end switch or when programmed to the safety distance (L2=).
- 8 The rotary table is positioned to its original position.
- 9 The calculated turning center displacement is corrected in the correction elements.
The difference between the old and new correction values is stored in E parameter (O1=, O2=).

When for D1 a value between -180 and +180 is given,

- The hole will be measured on 3 different positions of an arc. First on position C, after that on position C+D1:2 and latest on position C+D1.
- The table rotation centre will be calculated of the 3 centre points of the measured holes.
- When D1 equal 180 or +180 is, than the cycle sequence is equal to 2 measuring points.

Measuring result

The measuring results are written to a text file G640RESU.TXT at D:\startup.
In manual mode (MC320) a window is shown, e.g.:

Measured in:	[X]	[Y]
Offset old:	0.015	0.010
Rotat.center:	300.648	-480.043
Offset new:	0.010	0.012
Sum:	300.658	-480.031

Temperature:	22.3	
ESC = close information window		

Error messages

- P421 No correction element available
This error message appears when the relevant correction elements are not entered in the kinematics model.

Machine constants

MC843 Measuring feed rate [(μ m,mDeg)/min]
MC846 Measuring probe: orientation angles (0,1,2,3=all)
MC849 Measuring probe 1st orientation angle [Deg]

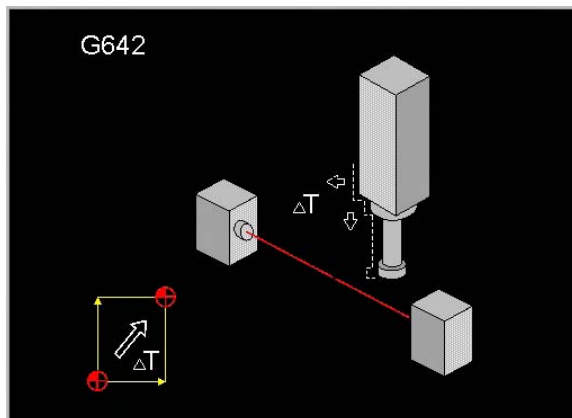
Example

N1 G17	set the surface plane
N2 T2 M6	Change the probe
N3 G0 X.. Y.. X..	Position the probe in the rotary table hole
N4 G640 C1=50 I1=1	Determine turning center
	The correction elements are corrected

9.15 G642 Laser: Temperature compensation

Measuring and correction of the spindle temperature expansion in 2 axes (HPC, Head Position Control) with the aid of a calibration tool and laser measuring system.

G642 corrects small temperature dependant axes errors. It corrects the radial axis (with respect to the laser), the tool axis and the head kinematics. An advantage is that the measurement is executed with rotating spindle so that the temperature remains stable.



```
G   Laser: temperature compensation
S   Speed
I2=  Suppress correction 0=no 1=yes
I3=  Textfile 0=overwrite 1=add
O1=  E-Par. temp. deviation rad. axis
O2=  E-Par. temp. deviation tool axis
```

O1=, O2= Output of the difference between the old and new correction values.

Basic settings

I2=0, I3=0

Notes and application

General

This cycle, used at higher accuracy demands, executes a temperature compensation for the NC-axes with the laser measuring system. The temperature dependant position change, mainly caused by the tool head, is compensated in the radial and axial axes and in the tool head. The errors occur because the automatic temperature compensation with sensor and correction table is calibrated for an average temperature development.

The cycle measures with the aid of a calibration tool the radial and axial positions of the laser beam. The difference with the calibrated laser position is stored in the kinematics chain machine constants to correct these axes.

Notes:

The incorporation of the temperature compensation measurement in the machining sequence should follow the schedule shown below:

- 1 Establish the turning center of the table with G640. Herewith the kinematics position of the table is corrected. For machine tools without rotary tables this measurement is skipped.
- 2 Next, calibrate with the calibration tool the laser measuring system (G600) to establish the actual machine kinematics as reference.
- 3 After this normal operation can take place: Measuring of the tools with the laser measuring system, setting the zero point by hand or with a measuring probe, work piece machining, etc.
- 4 Execute G642 regular. Depending on the thermal expansion of the machine tool and the required accuracy, the temperature compensation cycle can be executed before every n-th work piece or before a critical machining part.

Remark:

Measuring the kinematics and calibrating (item 1 and 2) is not required when the machine tool is switched on again in a batch production and the previous calibration is still valid.

Conditions:

- The measurement in the temperature compensation cycle G642 must be executed in vertical position. Doing so, the radial axis (in reference to the laser) and the tool axis are measured and corrected. The axis parallel to the laser beam **cannot** be corrected.
- The kinematics model of the machine tool must be entered and must contain correction elements for X, Y and Z. In case a rotary axis or swivel head in the tool head is present, also a correction element for the tool axis in the head must be available.
- The maximum correction per axis is $\pm 0.200\text{mm}$

Measuring result

The measuring results are written to a test file G642RESU.TXT at D:\startup, e.g.:

Temp	d-Rad	d-TI	Date	Time
22.3	0.013	0.034	10- 2-2003	10:05
22.4	0.014	0.036	10- 2-2003	10:06

Meaning:

Temp : Temperature of the sensors [°C].

d-Rad : Deviation, measured in the radial axis [mm|inch]

d-TI : Deviation measured in the tool axis [mm|inch]

Overwriting or adding the text file (I3=)

When during the cycle call overwrite is selected (I3=0), two lines, head and measuring data are re-written. When add (I3=1) is selected only one line with the measuring data is added. In this way a table is originated where the result of several measurements is visible.

Switching on:

The correction elements are set to zero when switching on the CNC.

Correction of the kinematics model

The deviation measured in the radial axis and tool axis, is corrected in the first correction element of the relevant axis from the table in the active kinematics model.

This correction element behaves like a zero point shift in the relevant axis.

The measures caused by swiveling are corrected separately via a correction element in the head. This measure is not directly measured, but is derived from the correction element in the table in the tool axis with the formula:

$$\text{head correction} = \text{total head correction} * \text{MC470} / 100,$$

where MC470: 'Temperature compensation: head lengthen/ distance [%]'.

Error message

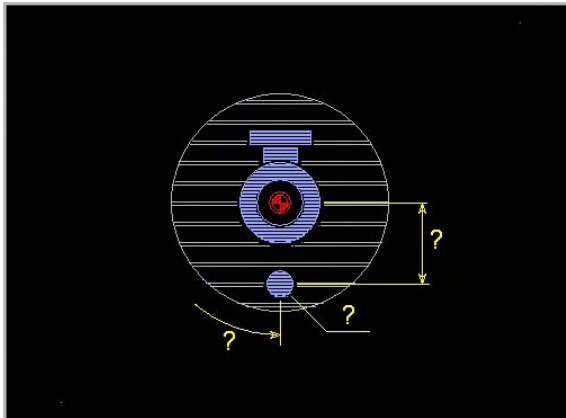
P421 No correction element available

This error message appears when the relevant correction elements are not entered in the kinematics model. When this happens, this G function cannot be used.

10. Specific cycles

G691 Measure unbalance
 G692 Unbalance checking
 G699 ATC (Application Tuning Cycle)

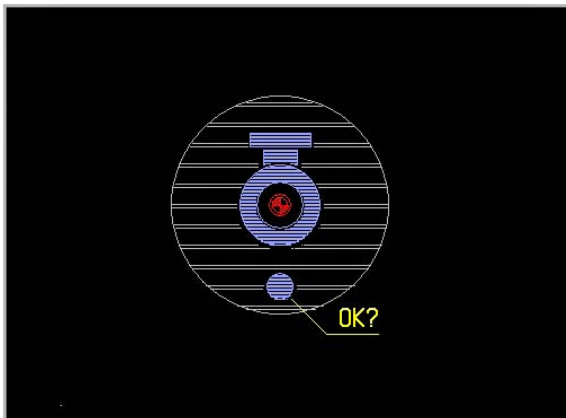
10.1 G691 Measure unbalance.



```
G  Unbalance measurement
D  Speed limitation      [rev/min]
```

For description see chapter: "Turning".

10.2 G692 Unbalance checking.

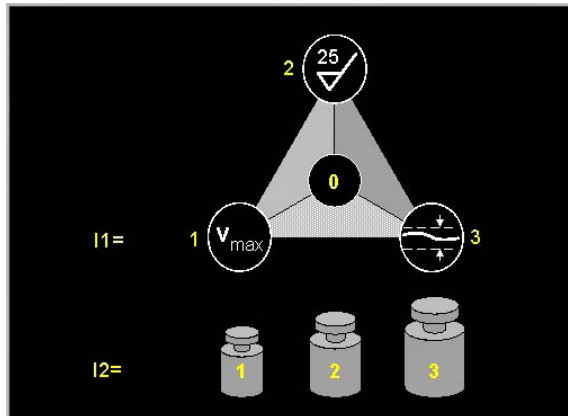


```
G  Unbalance check
C1= Allowed excursion   [mm|inch]
D  Check speed          [rev/min]
```

For description see chapter: "Turning".

10.3 G699 ATC- Cycle (= Application Tuning Cycle)

With the cycle G699 it is possible to optimise NC-programs depending on the execution stage. The utmost efficiency of the program is obtained with respect to the velocity, accuracy and surface quality.



```
G Application Tuning Cycle (ATC)
I1= 0=aus 1=schnell 2=Oberfl. 3=genau
I2= Masse 1=leicht 2=mittel 3=schwer
```

I1= Operation mode
 0 off.
 1 Velocity.
 2 Surface.
 3 Accuracy

I2= Mass
 1 Low (light workpiece).
 2 Medium (medium weight workpiece).
 3 Heavy (heavy workpiece).

Default settings

I1=0, I2=2

Notes and usage

General

Changing the operation mode to one of the values 0 to 3 adapts the movement behaviour. To this end internal control- and machine tool parameters are changed.

Weight mode.

Furthermore the influence of the workpiece weight is considered by means of a weight mode. Here extra adaptations to the relevant conditions can be made. However the maximum allowed table load must be taken into consideration.

For machine tools without a weight dependency in the axes the address I2 is not applicable.

Switching off

After "M30", <Cancel program> , <Clear control> or G699 I1=0 the default settings become active.

Display

When the ATC-cycle is active, it is shown in the dashboard.

Availability

Machine tool and CNC must be prepared for the ATC-cycle by the machine tool builder.

11. Machining and positioning cycles

The machining cycle defines a machining departure point. A separate positioning cycle defines execution of the machining cycle at a position.

11.1 Summary of machining and positioning cycles:

Special cycle:

1	G700	Facing	(only in DIN/ISO)
2	G730	Executing a pass	

Positioning cycles (Pattern)

(only in EASYoperate):

1	G771	Machining on a line	
2	G772	Machining on a rectangle	
3	G773	Machining on a grid	
4	G777	Machining on a circle	extension of G77
5	G779	Machining at a position	extension of G79

Drilling cycles:

1	G781	Drilling / centering	extension of G81
2	G782	Deep drilling	extension of G83
3	G783	Deep drilling (chip break)	extension of G83 (only in DIN/ISO)
4	G784	Tapping with compensating chuck	extension of G84 (only in EASYoperate)
5	G785	Reaming	extension of G85
6	G786	Hollow boring	extension of G86
7	G790	Reverse countersinking	
8	G794	Interpolating tapping	extension of G84 (only in EASYoperate)

Milling cycles:

1	G787	Pocket milling	extension of G87
2	G788	Slot milling	extension of G88
3	G789	Circular pocket milling	extension of G89
4	G797	Pocket finishing	
5	G798	Slot finishing	
6	G799	Circular pocket finishing	

11.2 Introduction

Machining plane

Cycle programming is independent of the machining plane (G17, G18, G19 and G7).

Tool axis and machining plane

The cycles are carried out in the current main plane G17, G18, G19 or in the inclined plane G7. The working direction of the cycle is determined by the tool axis. The direction of the tool axis can be reversed with G67.

Procedure in EASYoperate:

The machining cycles (special cycle, drilling cycle and milling cycle) are carried out on the patterns defined by the position cycles G77, G79, G771, G772, G773, G777 or G779.

General example:

Machining cycle (drilling cycle): N... G781
 Positioning cycle: N... G779 X... Y... Z...
 Cycle G781 is carried out in this position, determined by G779.

Procedure in DIN:

The new machining cycles (special cycle, drilling cycle and milling cycle) are only carried out by positioning cycle G79 in one position. Points (P1-P4) are not allowed.

Positioning logic

The tool moves in rapid motion, and depending on G28, using the positioning logic and the 1st setup clearance, to the position (X, Y, Z,) defined by the positioning cycle.

Mirroring and scaling

Mirroring and scaling are not allowed to be activated between a drilling/milling cycle and a positioning cycle.

Deleting cycle data

Cycle data is deleted by M30, the <Cancel program> softkey, the <Reset CNC> softkey or by defining a new cycle.

Switch on spindle

The spindle must be switched on for the cycle to start. F and S in the cycle definition can be overwritten.

Mirroring

If you are only mirroring one axis, the direction of rotation of the tool changes. This does not apply during machining cycles.

Comments

Comments are not allowed in a block with a machining cycle.
 Before calling up the cycle, you must program radius correction G40.

Warning

Pre-position the tool so that there can be no collision with the workpiece or clamping devices.

11.3 Description of addresses

Mandatory addresses

Mandatory addresses are shown in black. If a mandatory address is not entered an error message is issued.

Optional addresses

Optional addresses are shown in light grey. If these addresses are not entered they are ignored or given the basic setting that has already been entered.

Explanation of addresses.

The addresses described here are used in most cycles. Specific addresses are described in the cycle.

X, Y, Z: Position of the defined machining geometry

Machining is carried out in this position. If X, Y or Z is not entered, the current position of the tool is adopted.

Execution

The tool moves to the starting point in rapid motion and depending on G28, using positioning logic. If X, Y or Z is not programmed, the current position is taken as the starting point. The first setup clearance (L1=) is taken into account in the tool axis. When going down the lines (G730) the other axes are also displaced.

L Depth (greater than 0) When going down the lines (G730) this is the machining depth: distance between programmed workpiece surface and surface of unmachined part.

R Radius of the circular pocket

L1= 1st setup clearance at start of cycle.

L2= 2nd setup clearance: height above the 1st setup clearance.
At the end of the cycle the tool moves to the 2nd setup clearance (if entered).

C1= Feed depth (> 0): dimension used to adjust the tool each time. The depth (L) or machining depth (L) does not necessarily have to be a multiple of the feed depth (C1=). The CNC moves to the depth in one work pass if the feed depth is the same as or greater than the depth (C1=>L-L3).

Note:

If a feed depth (C1=) is programmed for milling or machining, there is usually a residual cut that is smaller than the programmed feed depth.

For drilling, the last 2 cuts are distributed equally if the residual cut >0. This avoids having a very small last cut.

D3= Dwell time: Number of revolutions for which the tool stays at the base of the hole for free cutting. (Minimum is 0 and maximum is 9.9.)

F2= Rapid plunging motion: traverse speed of tool when moving from setup clearance to the milling depth.

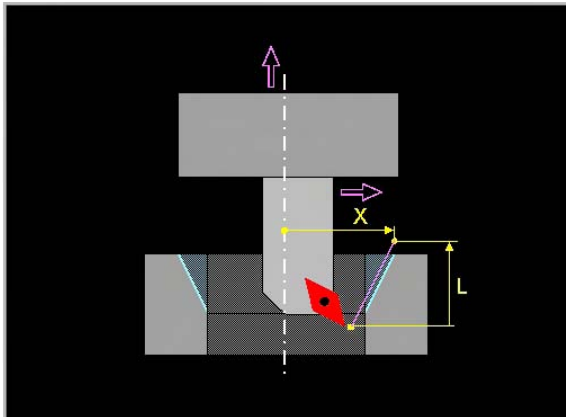
F5= Rapid retraction movement: traverse speed of tool when moving out of the hole.

F and S

The addresses F and S are not available in machining cycles within EASYoperate. They must be programmed in the FST menu.

11.4 G700 Facing cycle

Der Plandrehzyklus führt eine einzelne flache oder konische Drehbearbeitung aus.



```
G Face turning
X Radius
F2= Feed [mm/rev|inch/rev]
L Tool axis displacement
I1= Uncouple 0=no 1=yes
S Speed
```

Basic settings

L0, I1=0

EASYoperate ⇔ DIN/ISO

G700 is not available in EASYoperate.

The following addresses in the tool memory are used by the cycle:

- R Adjustment radius. Is automatically overwritten with the current radius after facing.
- A1 Orientation angle for engaging. Is automatically overwritten with the current angle (0-359.999 degrees) after facing.
- R1 Minimum diameter (optional)
- R2 Maximum diameter (optional)

Notes and application

G700 must not be programmed if:

- G36 and/or G182 are active.
- tool T0 is programmed.
- the spindle orientation at an angle is not allowed to be zero.

Resetting the radial facing slide:

The maximum speed allowed can be used to reset the radial facing slide to the starting diameter.

Actual diameter reached:

The programmed diameter is rounded so that it exactly matches one of the 72 indexing positions of the clamp. The maximum difference that this causes is $< (\text{feed}/72)/2$, i.e. 0.001mm deviation for 0.15mm feed/rev.

Note:

G40, G72, G90 and G94 remain active after G700

Block approach

In a block approach the head must be in the correct position before a G700 cycle starts. Therefore the radius R and angle A1 must be correctly entered in the tool table.

Speed and feed correction switch:

The speed correction switch is not active. The feed correction switch is active.

Display:

During movement the speed is displayed in the current S field. At the end the spindle position is always displayed in the range 0-359.999 degrees.

The programmed feed remains unchanged. The current feed displays zero or the feed of the traverse in the tool axis.

The cycle automatically indexes movement in and out:

M82 indexing of **outward** movement (in the facing head). M80 indexing of **inward** movement

Example:

Programming example	Description
	Tool memory: tool radius R20 Tool memory: orientation angle A1=0
N120 G700 X50 L5 F=0.05 S600	Chamfer 5mm from diameter 40 to 50
N140 G700 X70	Facing movement at diameter 70
N130 G0 Z100	Lift off
N140 G700 X40 I1=1 S1200	Return to diameter 40 and disengage

Facing head

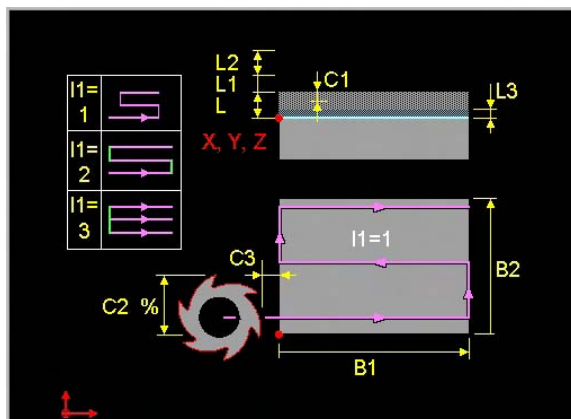
The facing head can be turned into the spindle and then used as a hollow boring head. The bracket is fixed by the indexing device built into the machine and at the same time the locking device between the bracket and facing head is loosened. When the spindle is rotating a mechanical gearing of e.g. 0.1mm per rev causes the radial facing slide to move. The transverse feed is determined by the rotary speed of the spindle. Synchronised movement of the spindle and tool axis (Z) enables cones and chamfers to be turned. Rotate the spindle anticlockwise to reset.

The cycle

- 1 Set the facing head adjustment radius and enter it into the tool memory.
- 2 Turn the facing head round in the spindle (the first time, check the engagement angle).
- 3 Check the orientation and indexing and run out if necessary.
- 4 The spindle turns, thus carrying out a facing movement.
- 5 Angle positions in multiples of 5 degrees are approached.
- 6 The adjustment radius and angle of orientation are automatically written into the tool memory

11.5 G730 Multipass milling

Define a single pass milling cycle in a single program block.



```
G    Multipass milling
B1=  1st Side length
B2=  2nd Side length
L    Height
L1=  1st Setup clearance
L2=  2nd Setup clearance
L3=  Finishing allowance
C1=  Plunging depth
C2=  Proportional cutting width
C3=  Radial setup clearance
I1=  1=meander 2=M.+rapid 3=parallel
F    Feed
S    Speed
F2=  Rapid for plunging
```

- B1= Length of 1st side in the main axis (with direction prefix)
 B2= Length of 2nd side in the secondary axis (with direction prefix)
 L= Machining height (>0)
 C2= Percentage cutting width: maximum percentage of the tool diameter to be used as the cutting width on each pass. The total width is divided into equal sections. On the last cut 10% of the diameter of the mill goes over the edge of the material.
 C3= radial setup clearance
 I1= Method:
 I1=1 Meander
 I1=2 meander and transverse movement out of the material
 I1=3 Machining in the same direction. The directions of B1= and B2= are used to determine whether to mill using forwards or reverse rotation.

The other addresses are described in the introduction to the machining cycles.

Basic settings

L1=1, L2=0, L3=0, C1=L-L3, C2=67%, C3=5, I1=1

The cycle

Method: meander

- 1 Rapid motion to the 1st setup clearance above the surface of the workpiece. The starting point is the radius of the tool plus the radial setup clearance (C3=) in addition to the programmed position.
- 2 Rapid plunging movement (F2=) by the feed depth (C1=) to the next depth.
- 3 After this the tool mills one line in the main axis. The end point of this movement is in the material by the cutting width (C2= maximum 50% of the milling cutter radius). In the last cut the tool travels outside the material by the amount of the radial clearance.
- 4 The tool moves with transverse milling advance to the starting point of the next pass. In the last pass it moves outside the material by 10% of the milling cutter radius.
- 5 Repeat steps 3 and 4 until all of the surface that has been defined has been machined.
- 6 Repeat steps 1 to 6 until the depth (L) has been reached.
- 7 At the end there is rapid movement to the 1st plus 2nd setup clearances (L1= plus L2=).

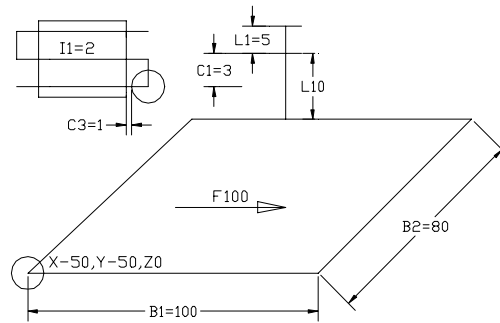
Method: meander and transverse movement out of the material

In this method the end point of each pass is outside the material by the amount of the radial setup clearance. The tool executes the transverse movement rapidly.

Method: milling in the same direction.

In this method the tool mills in the same direction on each pass (forward or reverse rotation). The end point of each pass is outside the material by the amount of the radial setup clearance. The CNC retracts the tool by the 1st setup clearance (L1=) at the end of a line. The tool then moves rapidly back to the main axis and then executes the transverse movement.

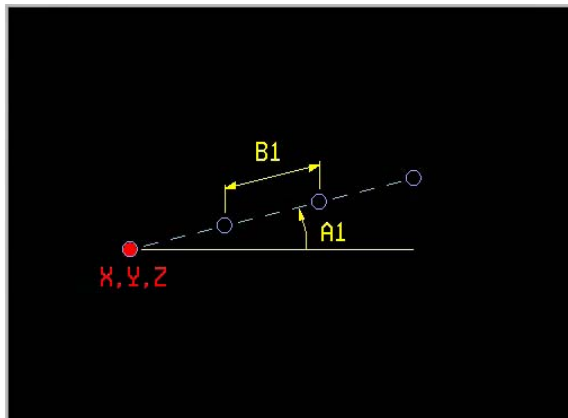
Example



Programming example	Description
N55 T1 M6	Change tool
N60 S500 M3	Switch on spindle
N65 G730 I1=2 B1=100 B2=80 L10 L1=5 C1=3 C2=73 C3=1 F100	Define multipass milling cycle
N70 G79 X-50 Y-50 Z0	Carry out multipass milling cycle

11.6 G771 Machining on a line

Execution of a machining cycle on points that are equally spaced out along a line.



G Operation on line
 X Position
 Y Position
 Z Position
 B1= Spacing
 K1= Number of operations
 A1= Angle
 F Feed

Basic settings

A1=0

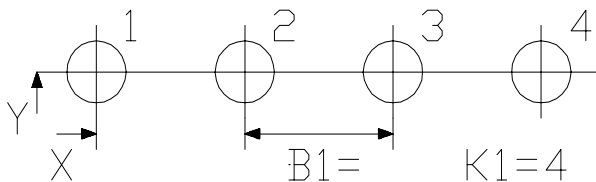
EASYoperate ⇔ DIN/ISO

G771 is only available in EASYoperate.

The cycle

1. Rapid movement into position.
2. The predefined machining cycle is executed at this point.
3. The tool then advances to the next position.
4. Repeat steps (2-3) until all positions (K1=) have been machined.

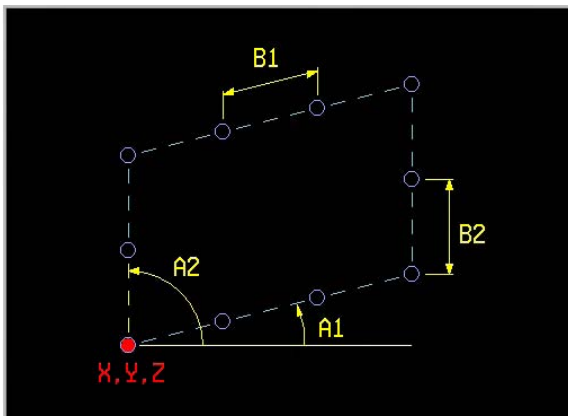
Example



Programming example	Description
N60 T1 M6	Change tool
N65 S500 M3	Switch on spindle
N70 G781 L-30 F100 F5=6000	Define drilling cycle
N75 G771 X50 Y20 Z0 B1=40 K1=4	Carry out drilling cycle at 4 points

11.7 G772 Machining on a rectangle

Execution of a machining cycle on points that are equally spaced out on a rectangle.



```
G  Operation on quadrangle
X  Position
Y  Position
Z  Position
B1= Longitudinal spacing
K1= Number of longitudinal operations
B2= Transverse spacing
K2= Number of transverse operations
A1= Starting angle
A2= Ending angle
F  Feed
```

Basic settings

A1=0, A2=90

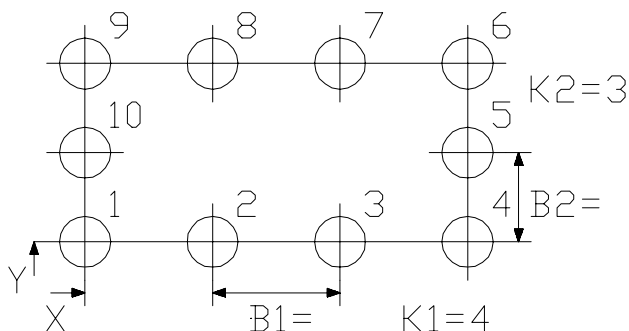
EASYoperate ⇔ DIN/ISO

G772 is only available in EASYoperate.

The cycle

1. Rapid movement into position.
2. The predefined machining cycle is executed at this point.
3. The tool then advances to the next position. The direction of the rectangle is determined by the angle A1=.
4. Repeat steps (2-3) until all positions (K1=, K2=) have been machined.

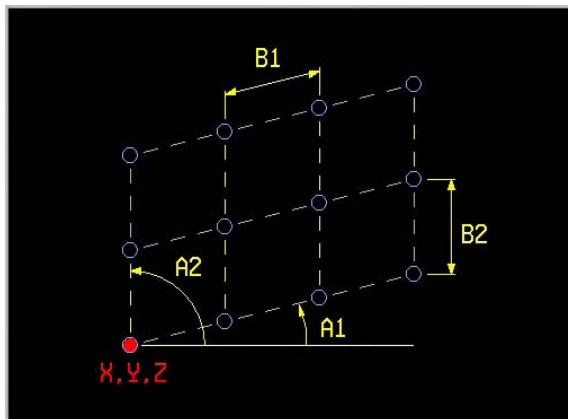
Example



Programming example	Description
N60 T1 M6	Change tool
N65 S500 M3	Switch on spindle
N70 G781 L-30 F100 F5=6000	Define drilling cycle
N75 G772 X50 Y20 Z0 B1=40 K1=4 B2=30 K2=3	Execute the drilling cycle at 10 points on the rectangle

11.8 G773 Machining on a grid

Execution of a machining cycle on points that are equally spaced out on a grid.



G Operation on grid
 X Position
 Y Position
 Z Position
 B1= Longitudinal spacing
 K1= Number of longitudinal operations
 B2= Transverse spacing
 K2= Number of transverse operations
 A1= Starting angle
 A2= Ending angle
 F Feed

Basic settings

A1=0, A2=90

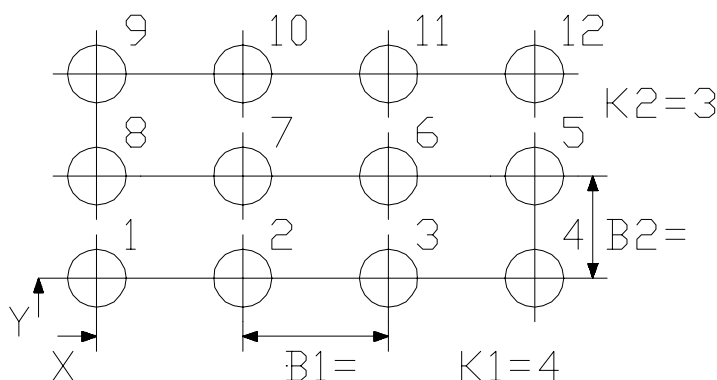
EASYoperate ↔ DIN/ISO

G773 is only available in EASYoperate.

The cycle

1. Rapid movement into position.
2. The predefined machining cycle is executed at this point.
3. The tool then advances to the next position. The tool advances in the initial direction to the positions using a zigzag movement, determined by the angle A1.
4. Repeat steps (2-3) until all positions (K1=, K2=) have been machined.

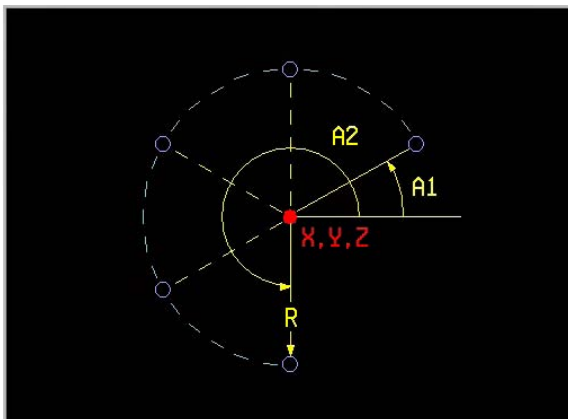
Example



Programming example	Description
N60 T1 M6	Insert tool 1
N65 S500 M3	Switch on spindle
N70 G781 L-30 F100 F5=6000	Define drilling cycle
N75 G773 X50 Y20 Z0 B1=40 K1=4 B2=30 K2=3	Execute the drilling cycle at 10 points on the grid

11.9 G777 Machining on a circle

Execution of a machining cycle on points that are equally spaced out on an arc or a full circle.



```
G  Operation on circle
X  Center position
Y  Center position
Z  Center position
R  Radius
K1= Number of operations
A1= Starting angle
A2= Ending angle
F  Feed
```

Basic settings

A1=0, A2=360

EASYoperate ⇔ DIN/ISO

G777 is only available in EASYoperate.

Note

Direction:

If A1= is greater than A2=, the holes are made clockwise.

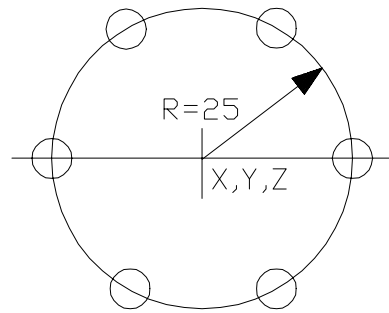
If A1= is less than or equal to A2=, the holes are made anticlockwise.

The cycle

1. Rapid movement into position.
2. The predefined machining cycle is executed at this point.
3. The tool then advances to the next position. The direction of the positions is determined by A1= and A2=.
4. Repeat steps (2-3) until all positions (K1=) have been machined.

Examples

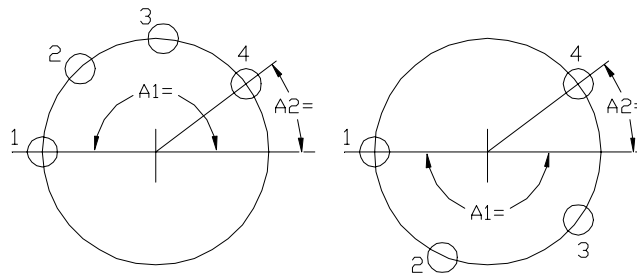
Example 1: Cycle on a full circle



Programming example	Description
N60 T1 M6	Change tool
N65 S500 M3	Switch on spindle
N70 G781 L-30 F100 F5=6000	Define drilling cycle
N75 G777 X50 Y20 Z0 R=25 K1=6 A1=0 A2=300	Execute the drilling cycle at 6 points on the circle K1=6 Number of holes =6 A1=0 Starting angle = 0 degrees A2=300 Stopping angle = 300 degrees
or	
N75 G777 X50 Y20 Z0 R=25 K1=7 A1=0, A2=360	Execute the drilling cycle at 6 points on the circle K1=7 Number of holes entered =7 Number of holes machined =6 A1=0 Starting angle = 0 degrees A2=360 Stopping angle = 300 degrees

Note: In this case 6 holes are drilled instead of 7, the number entered. The first and last holes in the cycle are in the same position. If an operation has to be carried out a second time in the same position during the cycle, the second operation is not executed.

Example 2 Direction of drilling on an arc



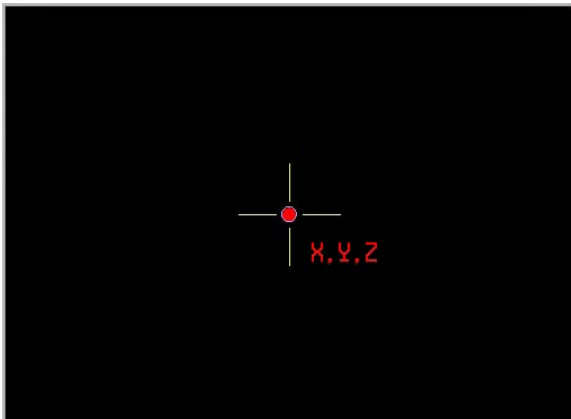
A1 = 180
A1 - A2 > 0 CW

A1 = -180
A1 - A2 < 0 CCW

Programming example	Description
N50 G81 Y1 Z-10 F100 S1000 M3	Define cycle
N60 G77 X0 Y0 Z0 R25 A1=180 A2=30 J4	Repeat the cycle four times on the arc; start at 180 degrees, end at 30 degrees going clockwise (CW).
N70 G77 X0 Y0 Z0 R25 A1=-180 A2=30 J4	Repeat the cycle four times on the arc; start at 180 degrees, end at 30 degrees going anticlockwise (CCW).

11.10 G779 Machining at a position

Ausführen eines Bearbeitungszyklus auf einer Position.



G Operation at position
X Position
Y Position
Z Position
F Feed

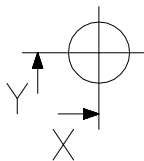
EASYoperate ⇔ DIN/ISO

G779 is only available in EASYoperate.

The cycle

1. Rapid movement into position.
2. The predefined machining cycle is executed at this point.

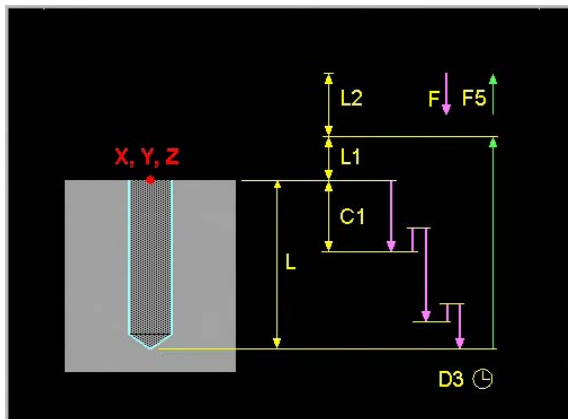
Example



Programming example	Description
N60 T1 M6	Change tool
N65 S500 M3	Switch on spindle
N70 G781 L-30 F100 F5=6000	Define drilling cycle
N75 G779 X50 Y20 Z0	Carry out drilling cycle at the point

11.11 G781 Drilling / centring

Define a simple drilling or centring cycle with possible chip break in a single program block.



```
G  Drilling / centring
L  Depth
L1= 1st Setup clearance
L2= 2nd Setup clearance
C1= Cutting depth
D3= Dwell [revolutions]
F  Feed
S  Spindle speed
F5= Retract rapid
```

Basic settings

L1=1, L2=0, C1=L, D3=0

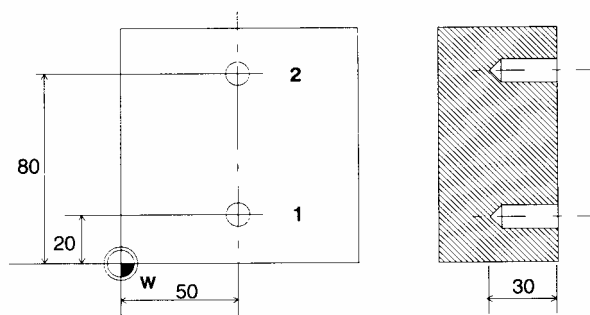
EASYoperate ⇔ DIN/ISO

The addresses D3=, F and S are not available in EASYoperate.

The cycle

1. Rapid motion to the 1st setup clearance (L1=).
2. Drilling with drilling advance by the cutting depth (C1=) or depth (L).
3. Rapid retraction (F5=) of 0.2mm
4. Repeat steps 2 to 3 until the drilling depth (L) has been reached.
5. At the bottom of the hole, dwell (D3=) for free cutting.
6. Rapid retraction (F5=) to 1st setup clearance (L1=) followed by rapid movement to 2nd setup clearance (L2=).

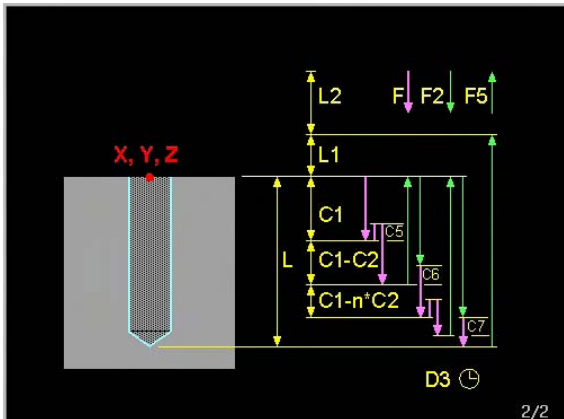
Example



Programming example	Description
N60 T1 M6	Change tool
N65 S500 M3	Switch on spindle
N70 G781 L30 F100 F5=6000	Define drilling cycle
N75 G79 X50 Y20 Z0	Carry out drilling cycle at point 1
N76 G79 X50 Y80 Z0	Carry out drilling cycle at point 2

11.12 G782 Deep hole drilling

Define a deep hole drilling cycle with reducing feed depth for chip break and regular chip removal in a single program block.



G Deep-hole drilling
 L Depth
 L1= 1st Setup clearance
 L2= 2nd Setup clearance
 C1= Cutting depth
 C2= Cutting depth reduction
 C3= Minimum cutting depth
 C5= Retract distance for chip break.
 C6= Safety distance after retract
 C7= Safety dist. after last retract
 K1= Number of steps before retract
 D3= Dwell [revolutions]
 F Feed
 S Spindle speed
 F2= In depth rapid

F5= Retract rapid

If the cutting depth (C1=) is not programmed or C1= is greater than or equal to the depth (L), the addresses C2=, C3=, C5=, C6=, C7= and K1= are meaningless.

If the number of steps to retraction (K1=) is not programmed or K1=1, the addresses C6= and C7= are meaningless.

With distributed cuts for chip break and/or chip removal.

- C2= Value by which the feed depth reduces after every advance. ($C1 = C1 - n * C2$). The feed depth (C1=) is always greater than or equal to the minimum feed depth (C3=).
- C5= Retraction distance for chip break (incremental): distance by which the tool retracts for chip breaking.

Chip removal after a number of cuts:

- K1= Number of advance movements (C1=) before the tool moves out of the hole for chip removal. For chip breaking without removal, the tool retracts each time by the retraction distance (C5=). If K1=0 chip removal takes not place.
- C6= Safety distance for rapid positioning when the tool returns to the current feed depth after being retracted from the hole. This value applies to the first advance.
- C7= Safety distance for rapid positioning when the tool returns to the current feed depth after being retracted from the hole. This value applies to the last advance.
 If C6= is not equal to C7=, the safety distance between the first and last cuts is gradually reduced.

The other addresses are described in the introduction to the machining cycles.

Basic settings

L1=1, L2=0, C1=L, C2=0, C3=C2, C5=0.1, C6=0.5, C7=0.5, K1=1, D3=0

EASYoperate ⇔ DIN/ISO

The addresses C5=, C6=, C7=, K1=, D3=, F and S are not available in EASYoperate.

Notes and application

Rules for distribution of cuts.

1. The cutting depth is always limited by the hole depth (L).
2. If C3 is programmed and there are 2 cuts, the first drilling cut can be reduced.
3. Every cut is smaller than or equal to the preceding one.
4. If there are more than 2 cuts plus a final cut, the final cut and the one preceding it are executed in 2 equal steps. This avoids having a very small final cut.

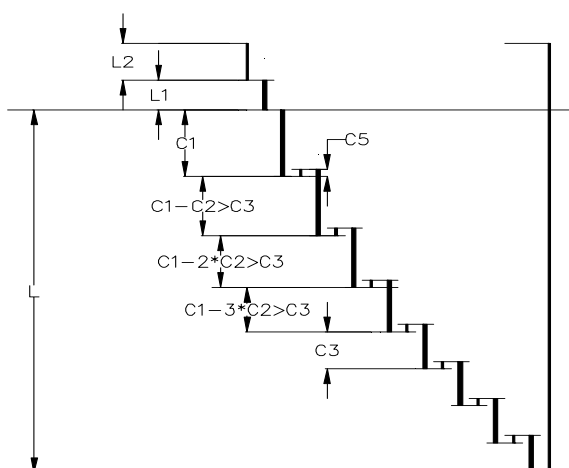
Examples of distribution of cuts.

Programming	Drilling cuts	Instructions or rules
One or two drilling cuts:		
G782 L10 C1=15	10	Rule 1
G782 L10 C1=9	9 1	
G782 L10 C1=9 C3=2	8 2	Rule 2
G782 L10 C1=7 C3=6	5 5	Rules 2 and 3
More than 2 drilling cuts		
G782 L25 C1=7	7 7 5.5 5.5	Rule 4
G782 L25 C1=7 C2=2	7 5 3 2 2 2 2 2	
G782 L24 C1=7 C2=2	7 5 3 2 2 2 1.5 1.5	Rule 4
G782 L29 C1=7 C2=2 C3=3	7 5 3 3 3 3 2.5 2.5	Rule 4

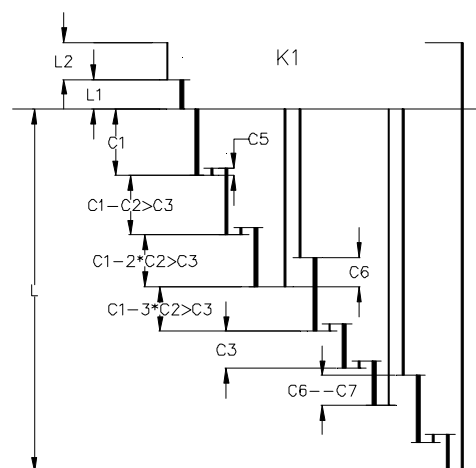
The cycle

- 1 Rapid motion to the 1st setup clearance (L1).
- 2 Drilling with drilling advance by the cutting depth (C1=).
- 3 For chip breaking: reverse movement by the retraction value (C5=).
- For chip removal: Rapid retraction (F5=) followed by rapid plunging (F2=) as far as the safety distance (C5= up, to C7= down).
- 4 The feed depth (C1=) then reduces by the cutting depth reduction (C2=). The minimum feed depth is equal to C3=.
- 5 Repeat steps 2 to 4 until the drilling depth (L) has been reached.
- 6 At the bottom of the hole, dwell (D3=) for free cutting.
- 7 Rapid retraction (F5=) to 1st setup clearance (L1=) followed by rapid movement to 2nd setup clearance (L2=).

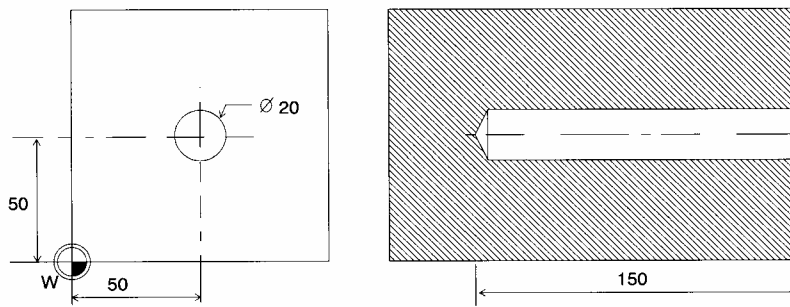
Machining sequence



Input: C1=..., K1=large



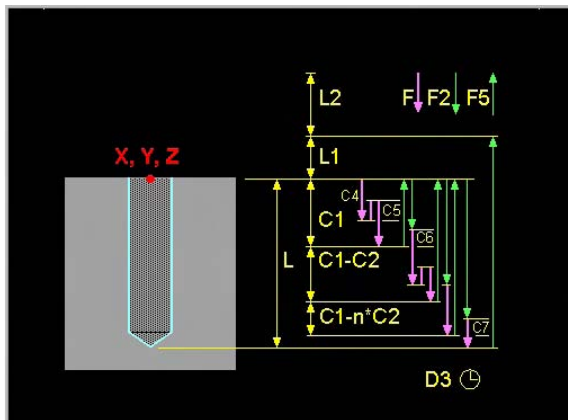
Input: C1=..., K1=3

Example

Programming example	Description
N5 T1 M6	Change tool
N10 S500 M3	Switch on spindle
N15 G782 L150 L1=4 C1=20 C2=3 C3=6	Define deep hole drilling cycle
N20 G79 X50 Y50 Z0	Execute deep hole drilling cycle

11.13 G783 Deep drilling (chip breaking)

Define a deep hole drilling cycle with reducing feed depth for chip removal and a fixed chip break distance in a single program block.



```
G   Deep-hole drill. add. chip break.
L   Depth
L1= 1st Setup clearance
L2= 2nd Setup clearance
C1= Cutting depth
C2= Cutting depth reduction
C3= Minimum cutting depth
C4= Drilling depth before chip break.
C5= Retract distance for chip break.
C6= Safety distance after retract
C7= Safety dist. after last retract
D3= Dwell [revolutions]
F   Feed
S   Spindle speed
F2= In depth rapid
```

```
F5= Retract rapid
```

If the cutting depth (C1=) is not programmed or C1= is greater than or equal to the depth (L), the addresses C2=, C3=, C4=, C5=, C6= and C7= are meaningless.

If the drilling depth before chip break (C4=) is not programmed or C4= is greater than or equal to the hole depth (L), the addresses C6= and C7= are meaningless.

C4= Advance after which a chip break is performed. If $C4 > C1$ or is not programmed there is no chip break.

C6= Safety distance for rapid positioning when the tool returns to the current feed depth after being retracted from the hole. This value applies to the first advance.

C7= Safety distance for rapid positioning when the tool returns to the current feed depth after being retracted from the hole. This value applies to the last advance.

If C6= is not equal to C7=, the safety distance between the first and last cuts is gradually reduced.

The other addresses are described in the introduction to the machining cycles.

Basic settings

L1=1, L2=0, C1=L, C2=0, C3=C1, C4=C1, C5=0.1, C6=0.5, C7=C6, D3=0

Notes

Cutting depth:

If more than 2 cuts are required the final cut and the one preceding it are executed in 2 equal steps.

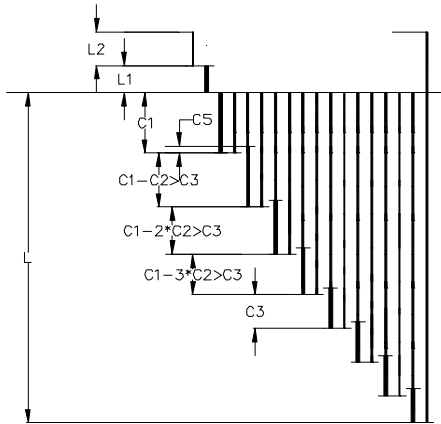
This avoids having a very small final cut.

The cycle

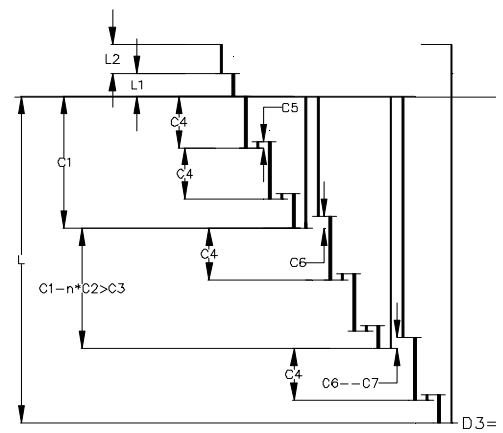
- 1 Rapid motion to the 1st setup clearance.
- 2 No chip break ($C4 > C1$ or C4 not programmed: drilling with drilling advance by the cutting depth (C1=).
With chip break ($0 < C4 < C1$): drill to depth (C4=). After this, retract by the retraction distance (C5=). Repeat until the cutting depth (C1=) is reached.
- 3 Rapid retraction (F5=) followed by rapid plunging (F2=) as far as the safety distance (C5= up, to C7= down).
- 4 The feed depth (C1=) then reduces by the cutting depth reduction (C2=). The minimum feed depth is equal to C3=.
- 5 Repeat steps 2 to 4 until the drilling depth (L) has been reached.

- 6 At the bottom of the hole, dwell (D3=) for free cutting.
- 7 Rapid retraction (F5=) to 1st setup clearance (L1=) followed by rapid movement to 2nd setup clearance (L2=).

Machining sequence

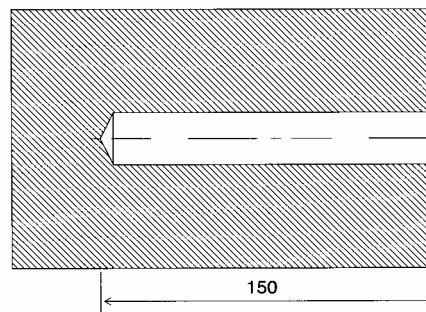
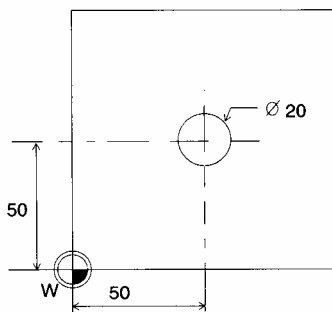


Input: C1=., C4=C1



Input: C1=., C4<C1

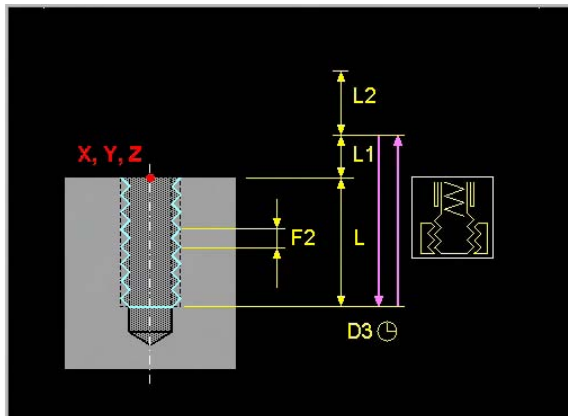
Example



Programming example	Description
N5 T1 M6	Change tool
N10 S500 M3	Switch on spindle
N15 G783 L150 L1=4 C1=20 C4=5 C2=2 C3=6 C5=0.5 F200	Define deep hole drilling cycle
N20 G79 X50 Y50 Z0	Execute deep hole drilling cycle

11.14 G784 Tapping with compensating chuck

Define a tapping cycle in a single program block.



```
G    Tapping
L    Depth
F2=  Pitch
L1=  1st Setup clearance
L2=  2nd Setup clearance
D3=  Dwell time [s]
```

L Depth (> 0)
 L1= Guideline value: 4x pitch
 D3= Length of time in seconds that the tool dwells at the bottom of the hole.

Basic settings

L1=1, L2=0, D3=0

EASYoperate ↔ DIN/ISO

G784 is only available in EASYoperate.

Notes and application:

The tool must be clamped in a linear compensation chuck. A linear compensation chuck compensates for the advance and speed tolerances during machining.

At the end of the cycle the coolant and spindle are restored to their status before the cycle.

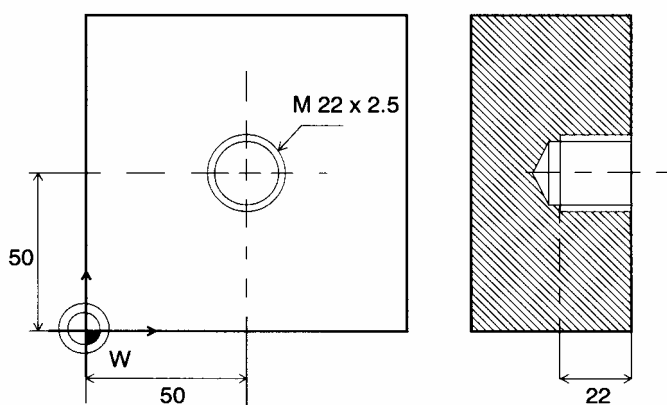
The advance is determined by the speed. Speed override is active during tapping. Feed override is not active.

When a G784 cycle is called up using G79 the CNC must be set to G94 mode (advance in mm/min), not G95 (advance in mm/rev).

Machine and CNC must be prepared for the G784 cycle by the machine builder.

The cycle

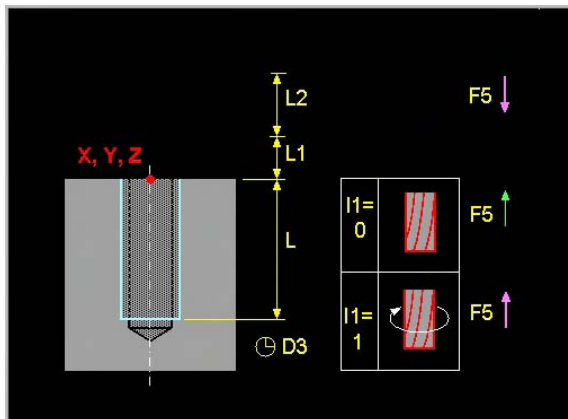
1. Rapid motion in the spindle axis to the 1st setup clearance (L1=).
2. Tapping with pitch (L3=) to depth (L).
3. After the dwell time (D3=) the direction of spindle rotation is reversed.
4. The tool is retracted with the pitch (L3=) to the 1st setup clearance (L1=) and then rapidly retracted to the 2nd setup clearance (L2=).
5. At the end the direction of spindle rotation is reversed once more.

Example

Programming example	Description
N13 T3 M6	Insert tool 3
N14 S56 M3	Switch on spindle
N15 G784 L22 L1=9 L3=2.5	Define the tapping cycle A linear compensation chuck must be used.
N20 G79 X50 Y50 Z0	Execute the cycle at the programmed position

11.15 G785 Reaming

Define a single pass reaming cycle in a single program block.



```
G Reaming
L Depth
L1= 1st Setup clearance
L2= 2nd Setup clearance
I1= Spindelstop 0=yes 1=no
D3= Dwell [revolutions]
F Feed
S Spindle speed
F5= Retract rapid
```

- I1= 0: Retraction with rapid movement and stationary spindle
 1: Retraction with advance and rotating spindle
 F5= Rapid movement (I1=0) or advance (I1=1) retraction: Traverse speed of tool when moving out of the hole in mm/min.

The other addresses are described in the introduction to the machining cycles.

Basic settings

L1=1, L2=0, I1=0, D3=0

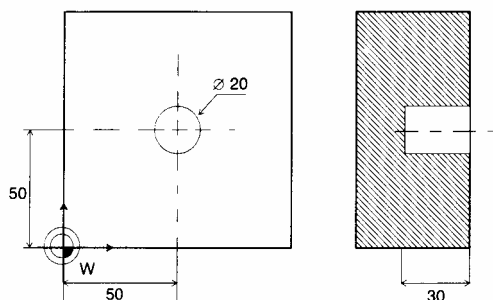
EASYoperate ⇔ DIN/ISO

The addresses D3=, F and S are not available in EASYoperate.

The cycle

- 1 Rapid motion to the 1st setup clearance (L1=).
- 2 Reaming with advance F down to depth (L).
- 3 At the bottom of the hole, dwell (D3=).
- 4 Rapid retraction (F5=) to 1st setup clearance (L1=) followed by rapid movement to 2nd setup clearance (L2=).

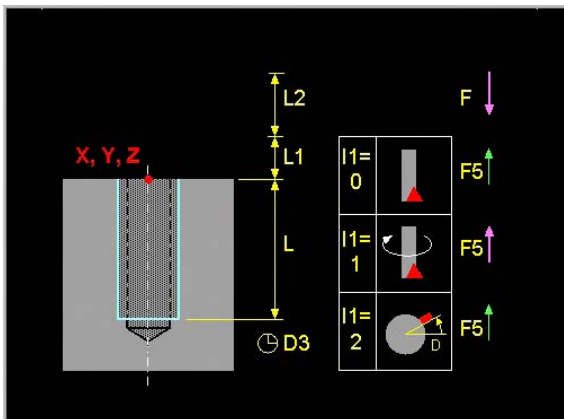
Example



Programming example	Description
N25 T4 M6	Change tool
N30 S1000 M3	Switch on spindle
N35 G785 L29 D3=2 F100 F5=2000	Define reaming cycle
N34 G79 X50 Y50 Z0	Execute the reaming cycle at the programmed position

11.16 G786 Boring

Define a cycle with the option to move clear with an oriented spindle in a single program block.



```
G Boring
L Depth
L1= 1st Setup clearance
L2= 2nd Setup clearance
C1= Retract distance from side
D Orientation angle tool tip
D3= Dwell [revolutions]
I1= Retract 0=M5 1=M3/M4 2=M19
F Feed
S Spindle speed
F5= Retract rapid
```

- C1= Distance by which the tool is retracted from the wall when moving clear.
 I1= 0: retract with rapid movement and stationary spindle without moving clear.
 1: retract with advance movement and rotating spindle without moving clear.
 2: with oriented spindle (M19) and rapid retraction.
 D Angle (absolute) at which the tool positions itself before moving clear (I1=2 only). The direction of moving clear is -X in G17/G18 and -Y in G19.
 F5= Rapid movement (I1=0 or I1=2) or advance (I1=1) retraction: Traverse speed of tool when moving out of the hole in mm/min.
 The other addresses are described in the introduction to the machining cycles.

Basic settings

L1=1, L2=0, C1=0.2, D=0, D3=0, I1=0, F5=rapid motion (I1=0 or I1=2) or F5=F (I1=1)

Notes and application

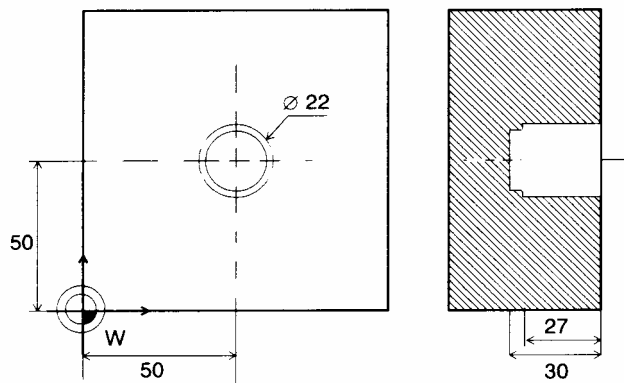
At the end of the cycle the spindle status that was active before the cycle is reactivated.

Risk of collision

The direction of the tool tip (MDI) should be such that it points to the positive main axis. The angle displayed should be entered as the orientation angle (D) so that the tool moves away from the edge of the hole in the direction of the negative main axis. The direction of moving clear is -X in G17/G18 and -Y in G19.

The cycle

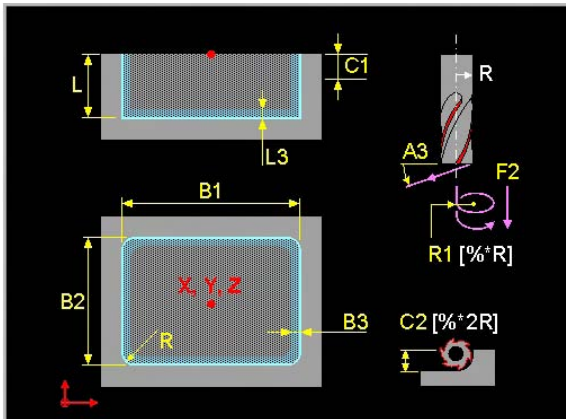
- 1 Rapid motion to the 1st setup clearance (L1=).
- 2 Reverse boring with advance (F) down to depth (L).
- 3 At the bottom of the hole, dwell (D3=) with running spindle for free cutting.
- 4 With I1=2 there is spindle orientation (D=) and a reverse movement along the main axis to the retraction distance (C1=).
- 5 Rapid retraction (F5=) to 1st setup clearance (L1=) followed by rapid movement to 2nd setup clearance (L2=).

Example

Programming example	Description
N45 T5 M6	Change tool
N50 S500 M3	Switch on spindle
N55 G786 L27 L1=4 L2=10 D3=1 F100	Define reverse boring cycle
N60 G79 X50 Y50 Z0	Execute the cycle at the programmed position

11.17 G787 Pocket milling

Define a pocket milling cycle for rough machining of rectangular pockets in a single program block. This cycle allows oblique plunging and mills in a continuous spiral path.



```
G   Pocket milling
B1=  1st Side length
B2=  2nd Side length
L   Depth
L1=  1st Setup clearance
L2=  2nd Setup clearance
L3=  Finishing allowance bottom
B3=  Finishing allowance sides
C1=  Plunging depth
C2=  Proportional cutting width
R   Rounding radius
R1=  Proportional helix radius
A3=  Plunging angle
I1=  Milling 1=climb -1=conventional
F   Feed
```

```
S   Speed
F2=  Feed for plunging
```

- B1= Length of the pocket in the main axis.
- B2= Width of the pockets in the secondary axis.
- C2= Percentage of the tool diameter to be used as the cutting width on each pass. The total width is divided into equal sections.
- R Radius for the corners of the pocket. Where radius $R=0$, the rounding radius is the same as the tool radius.
- R1= Percentage of the tool diameter to be used as the cutting width (>0) on oblique plunging.
- A3= Angle (0 to 90°) at which the tool can plunge into the workpiece. The plunging angle is adjusted so that the tool always plunges with a whole number of rectangular movements. It only plunges vertically at 90° .

The other addresses are described in the introduction to the machining cycles.

Basic settings

$L1=1$, $L2=0$, $L3=0$, $B3=0$, $C1=L$, $C2=67\%$, R = tool radius, $R1=80\%$, $A3=90^\circ$, $I1=1$, $F2=0.5 \cdot F$ for vertical plunging $F2=F$ for oblique plunging.

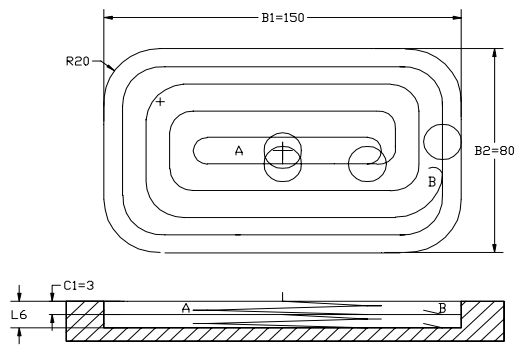
Notes and application

$B1$ = and $B2$ = must be greater than $2 \cdot (\text{tool radius} + \text{finishing allowance for sides } B3)$.

For finishing, the dimensions $L3$ and $B3$ must be entered.

The cycle

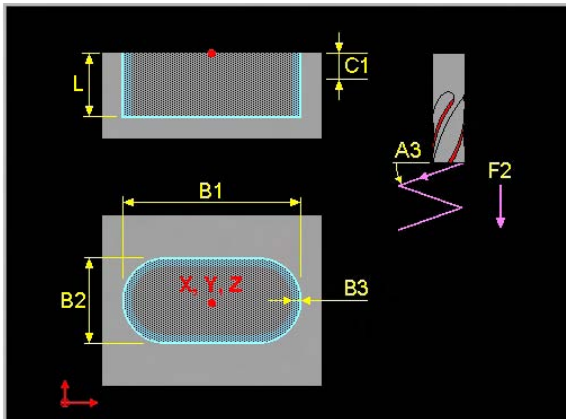
- 1 Rapid motion to the 1st setup clearance ($L1$) above the centre of the pocket.
- 2 If the plunging angle $A3=90^\circ$, the tool advances with feed ($F2$) to the first feed depth ($C1$).
If the plunging angle $A3<90^\circ$, the tool advances obliquely, using a whole number of rectangular movements, to the first feed depth ($C1$) with plunging feed ($F2$).
- 3 Machining with feed (F) in the positive direction of the long side, in a flowing movement from inside to outside.
- 4 At the end of this process the tool is retracted from the wall and the floor in a tangent to the helix and brought rapidly to the centre.
- 5 Repeat steps 2 to 4 until the depth (L) has been reached.
- 6 At the end there is rapid movement to the 1st plus 2nd setup clearances ($L1$ plus $L2$).

Example

Programming example	Description
N10 T1 M6 (R8 milling cutter)	Change tool
N20 S500 M3	Switch on spindle
N30 G787 B1=150 B2=80 L6 L1=1 A3=5 C1=3 C2=60 R20 I1=1 F200	Define pocket milling cycle
N40 G79 X160 Y120 Z0	Execute the cycle at the programmed position

11.18 G788 Key-way milling

Define a pocket milling cycle for rough machining and/or finishing of a slot in a single program block. This cycle allows oblique plunging.



```
G   Key-way milling
B1=  1st Side length
B2=  2nd Side length
L   Depth
L1=  1st Setup clearance
L2=  2nd Setup clearance
B3=  Finishing allowance sides
C1=  Plunging depth roughing
A3=  Plunging angle
I1=  Milling 1=climb -1=conventional
I2=  0=roughing 1=roughing + finishing
F   Feed
S   Speed
F2=  Feed for plunging
```

- B1= Length of slot in the main axis
 B2= Width of the slot in the secondary axis. If the slot width is the same as the tool diameter it is only roughed.
 A3= Maximum angle (0 to 90°) at which the tool can plunge into the workpiece. It only plunges vertically at 90°.
 I2= 0: Roughing only.
 1: Roughing and finishing.

The other addresses are described in the introduction to the machining cycles.

Basic settings

L1=1, L2=0, B3=0, C1=L, A3=90, I1=1, I2=0, F2=0.5*F for vertical plunging and F2=F for oblique plunging.

Notes and application

- When roughing with oblique plunging, there is a pendulum effect as the tool plunges into the material from one end of the slot to the other. There is thus no need to pre-drill.
- Vertical plunging always takes place into the end of the slot on the negative side. Pre-drilling is required at this point.
- Choose a milling cutter whose diameter is no greater than the width of the slot and no smaller than a third of the slot width.
- The diameter of the milling cutter chosen must be less than half the length of the slot, otherwise the CNC cannot use the pendulum effect for plunging.
- For finishing the dimension (B3=) must be entered.

The cycle

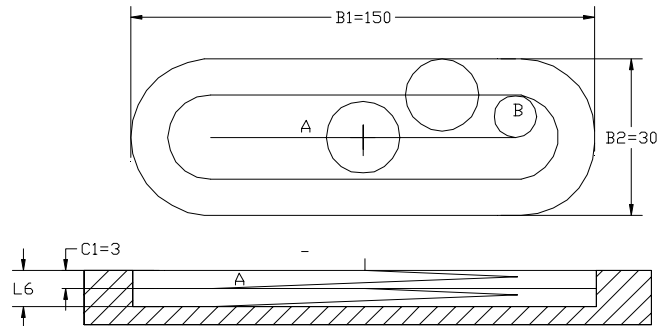
Roughing:

1. Rapid motion to the 1st setup clearance (L1=) and into the centre of the left circle.
2. If the plunging angle A3=90°, the tool advances with feed (F2=) to the first feed depth (C1=) and then with feed F into the centre of the right circle.
 If the plunging angle A3<90°, the tool advances obliquely, with plunging feed (F2=), using oblique motion, into the centre of the right circle. The tool then moves back to the centre of the left circle, again plunging obliquely. These steps are repeated until the cutting depth (C1=) is reached.
3. At the milling depth, the tool moves to the other end of the slot and then machines the slot shape until the finishing dimension is reached.
4. Repeat steps 2 to 3 until the programmed depth (L) has been reached.

Finishing:

5. The tool moves tangentially in the left or right circle of the slot at the contour and finishes it in forwards rotation ($I1=1$).
6. At the end of the contour the tool moves tangentially away from the contour and floor to the centre of the slot.
7. At the end there is rapid movement to the 1st plus 2nd setup clearances ($L1=$ plus $L2=$).

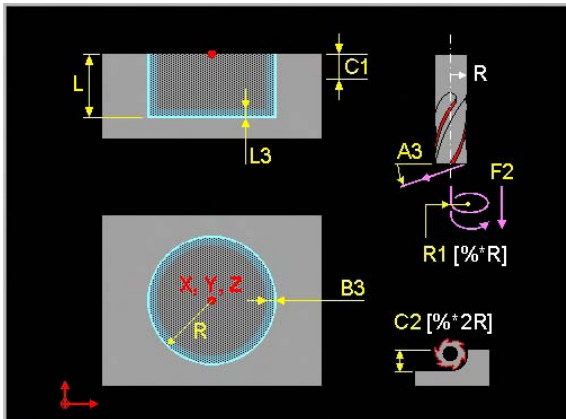
Example



Programming example	Description
N10 T1 M6 (R10 milling cutter)	Change tool
N15 S500 M3	Switch on spindle
N20 G788 B1=150 B2=30 L6 L1=1 A3=5 C1=3 I1=1 I2=0 F200	Define the slot milling cycle, parallel to the X axis
N30 G79 X20 Y20 Z0	Execute the cycle at the programmed position

11.19 G789 Circular pocket milling

Define a pocket milling cycle for rough machining of circular pockets in a single program block. This cycle allows oblique plunging and mills a continuous spiral path.



```
G  Circular pocket milling
R  Radius
L  Depth
L1= 1st Setup clearance
L2= 2nd Setup clearance
L3= Finishing allowance bottom
B3= Finishing allowance sides
C1= Plunging depth
C2= Proportional cutting width
R1= Proportional helix radius
A3= Plunging angle
I1= Milling 1=climb -1=conventional
F  Feed
S  Speed
F2= Feed for plunging
```

- C2= Percentage of the tool diameter to be used as the cutting width on each pass. The total width is divided into equal sections.
- R1= Percentage of the tool diameter to be used as the cutting width (>0) on oblique plunging.
- A3= Angle (0 to 90°) at which the tool can plunge into the workpiece. It only plunges vertically at 90°.

The other addresses are described in the introduction to the machining cycles.

Basic settings

L1=1, L2=0, L3=0, B3=0, C1=L, C2=67%, R1=80%, A3=90, I1=1, F2=0.5*F for vertical plunging and F2=F for oblique plunging.

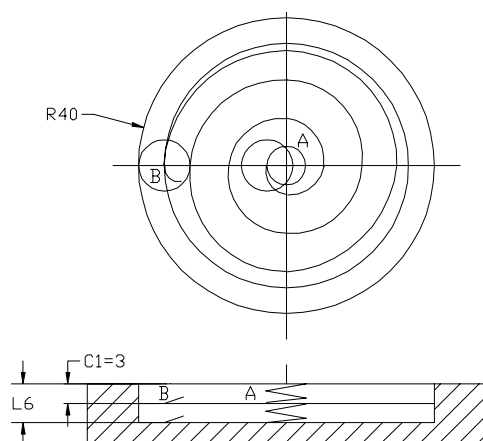
Notes and application

R must be greater than 2*(tool radius + finishing allowance for sides B3=).

For finishing, the dimensions L3 and B3 must be entered.

The cycle

1. Rapid motion to the 1st setup clearance (L1=) above the centre of the pocket.
2. If the plunging angle A3=90°, the tool advances with feed (F2=) to the first feed depth (C1=).
If the plunging angle A3<90°, the tool advances obliquely with plunging feed (F2=), using a number of circular movements, to the first feed depth (C1=).
3. Machining with feed (F) in an outwards-moving spiral.
4. At the end of this process the tool is retracted from the wall and the floor in a tangent to the helix and brought rapidly to the centre.
5. Repeat steps 2 to 4 until the depth (L) has been reached.
6. At the end there is rapid movement to the 1st plus 2nd setup clearances (L1= plus L2=).

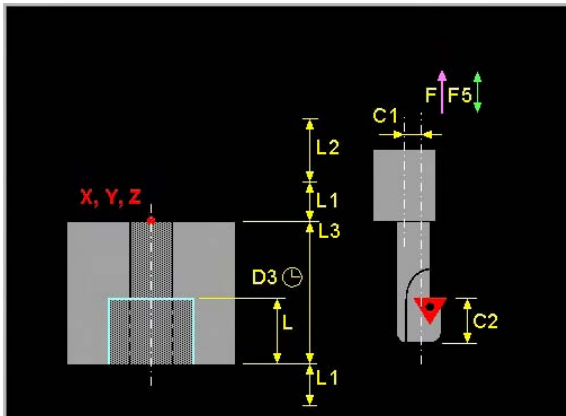
Example

Programming example	Description
N10 T1 M6 (R8 milling cutter)	Change tool
N20 S500 M3	Switch on spindle
N30 G789 R40 L=6 L1=1 A3=5 C1=3 C2=65 I1=1 F200	Define pocket milling cycle
N40 G79 X160 Y120 Z0	Execute the cycle at the programmed position

11.20 G790 Back-boring

Define a reverse countersinking cycle in a single program block.

The cycle only operates with reverse boring bars to create countersinks on the underside of the workpiece.



G Back-boring
 L Counterbore depth
 L3= Material thickness
 C1= Eccentricity
 L1= 1st Setup clearance
 L2= 2nd Setup clearance
 C2= Cutting edge height
 D Orientation angle tool tip
 D3= Dwell [revolutions]
 F Feed
 S Spindle speed
 F5= Retract rapid

L3= Thickness of workpiece

C1= Eccentricity of the boring bar (to be taken from the tool data sheet)

C2= Distance from bottom edge of boring bar to main cutter (to be taken from the tool data sheet)

D Angle (absolute) at which the tool positions itself before plunging and before moving out of the hole. The direction of moving clear is -X in G17/G18 and -Y in G19.

The other addresses are described in the introduction to the machining cycles.

Basic settings

L1=1, L2=0, C2=0, D=0, D3=0.2, F5=rapid motion

Notes and application

Enter the tool length so that the cutting edge of the boring bar is dimensioned.

The CNC takes the height of the cutting edge (C2=) into account when calculating the starting point.

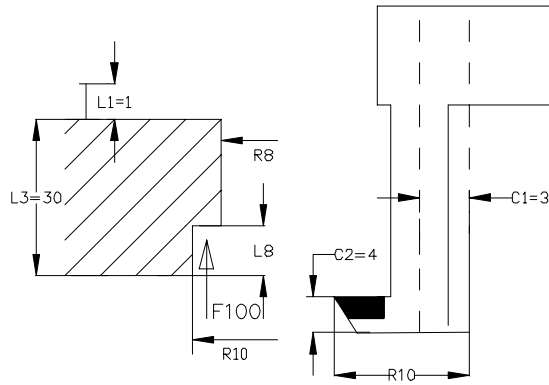
At the end of the cycle the spindle status that was active before the cycle was called up is reactivated.

Risk of collision

The direction of the tool tip (MDI) should be such that it points to the positive main axis. The angle displayed should be entered as the orientation angle (D) so that the tool moves away from the edge of the hole in the direction of the negative main axis. The direction of moving clear is -X in G17/G18 and -Y in G19.

The cycle

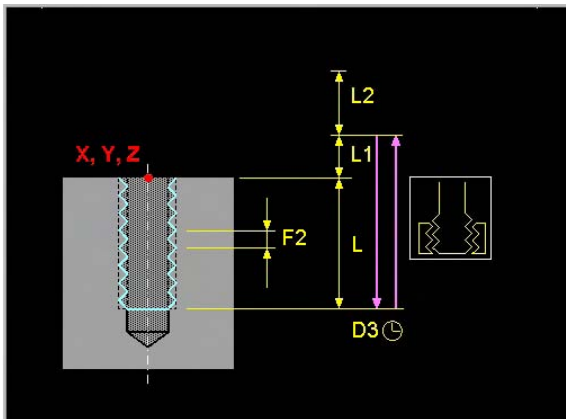
- 1 Rapid motion to the 1st setup clearance (L1=).
- 2 Spindle orientation to the D position and tool offset by the eccentricity dimension (C1=).
- 3 Rapid retract (F5=) plunging into the pre-drilled hole until the cutting edge is at the 1st setup clearance (L1=) below the bottom of the workpiece.
- 4 Movement to the centre of the hole, switch on spindle and coolant and machine at countersinking feed to the depth that has been entered.
- 5 At the bottom of the hole, the tool dwells with running spindle for free cutting.
- 6 The tool then moves out of the hole, performs spindle orientation and is once again displaced by the eccentricity dimension (C1=).
- 7 At the end, rapid retraction (F5=) to 1st setup clearance (L1=) followed by rapid movement to 2nd setup clearance (L2=).

Example

Programming example	Description
N60 T1 M6	Change tool (Tool radius R10, eccentricity C1=3, cutting edge height C2=4, angle for spindle orientation D0)
N65 S500 M3	Switch on spindle
N70 G790 L3=30 L8 L1=1 C1=3 C2=4 F100	Define reverse countersinking cycle
N75 G79 X30 Y40 Z0	Carry out defined cycle at the point

11.21 G794 Interpolated tapping

Define a tapping cycle with interpolation in a single program block.



```
G Tapping, interpolated
L Depth
F2= Pitch
L1= 1st Setup clearance
L2= 2nd Setup clearance
```

Basic settings

L1=1, L2=0

EASYoperate ⇔ DIN/ISO

G794 is only available in EASYoperate.

Notes and application:

At the end of the cycle the coolant status and spindle status that were active before the cycle are reactivated.

The advance is determined by the speed. Speed override is active during tapping. Feed override is not active.

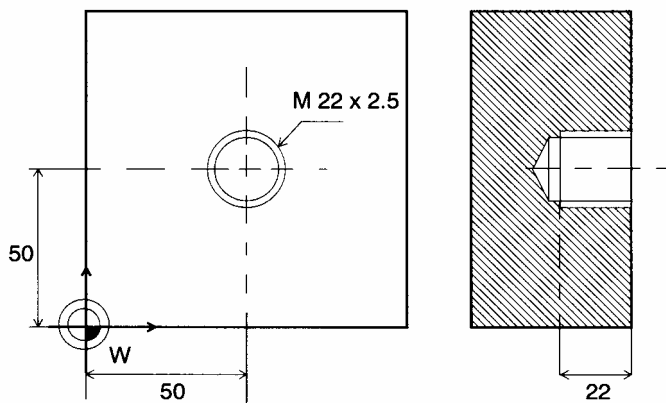
When a G794 cycle is called up using G79 the CNC must be set to G94 mode (advance in mm/min).

The spindle machine constants for interpolation should be correctly set during tapping. The spindle acceleration for each gear is calculated using MC2491, 2521, 2551, 2581 and MC2495, 2525, 2555, 2585. MC4430 should also be active in all cases to ensure proper adjustment.

Machine and CNC must be prepared for the G794 cycle by the machine builder.

The cycle

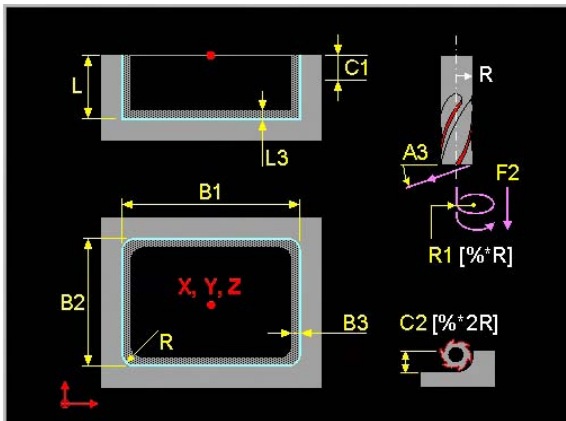
- 1 Rapid motion in the spindle axis to the 1st setup clearance (L1=) and spindle orientation once there.
- 2 Tapping with pitch (L3=) to depth (L).
- 3 The direction of spindle rotation is then reversed once more.
- 4 The tool is retracted with the pitch (L3=) to the 1st setup clearance (L1=) and then rapidly retracted to the 2nd setup clearance (L2=).
- 5 The spindle is stopped here.

Example

Programming example	Description
N13 T3 M6	Insert tool 3
N14 S56 M3	Switch on spindle
N15 G794 L22 L1=9 L3=2.5	Define the tapping cycle
N20 G79 X50 Y50 Z0	Execute the cycle at the programmed position

11.22 G797 Pocket finishing

Define a rectangular pocket milling cycle for finishing the wall and floor of rectangular pockets in a single program block. The sides can be machined in a number of advances. This cycle allows oblique plunging into the floor and mills in a continuous spiral path.



G Pocket finishing
 B1= 1st Side length
 B2= 2nd Side length
 L Depth
 L1= 1st Setup clearance
 L2= 2nd Setup clearance
 L3= Allowance bottom
 B3= Allowance sides
 C1= Plunging depth
 C2= Proportional cutting width
 R Rounding radius
 R1= Proportional helix radius
 A3= Plunging angle
 I1= Milling 1=climb -1=conventional
 I2= Finishing 0=complete 1=sides

F Feed
 S Speed
 F2= Feed for plunging

- B1= Length of the pocket in the main axis.
 B2= Width of the pocket in the secondary axis
 B3= Allowance sides, which will be removed by finishing.
 L3= Allowance bottom, which will be removed by finishing.
 C2= Percentage of the tool diameter to be used as the cutting width on each pass. The total width is divided into equal sections.
 R Radius for the corners of the pocket. Where radius $R=0$, the rounding radius is the same as the tool radius.
 R1= Percentage of the tool diameter to be used as the helix radius (>0) on oblique plunging.
 A3= Angle (0 to 90°) at which the tool can plunge into the workpiece. The plunging angle is adjusted so that the tool always plunges with a whole number of rectangular movements. It only plunges vertically at 90° .
 I2= 0: Finishing wall and floor
 1: Finish machining of wall only

The other addresses are described in the introduction to the machining cycles.

Basic settings

$L1=1$, $L2=0$, $L3=0$, $B3=1$, $C1=L$, $C2=67\%$, R = tool radius, 0 , $R1=80\%$, $A3=90^\circ$, $I1=1$, $F2=0.5 \cdot F$ for vertical plunging and $F2=F$ for oblique plunging.

Notes and application

$B1=$ or $B2=$ must be greater than $2 \cdot (\text{tool radius} + \text{finishing allowance for sides } B3)$.

The cycle

- 1 Rapid motion to the 1st setup clearance ($L1=$) above the centre of the pocket.

Finishing the floor:

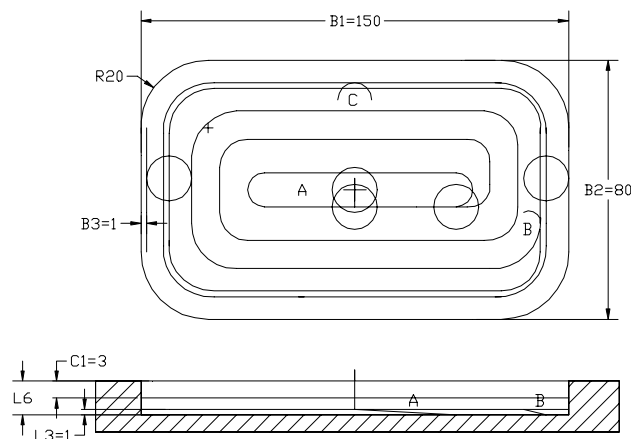
- 2 If the plunging angle $A3=90^\circ$, the tool advances with drilling feed ($F2=$) to the depth (L).
 If the plunging angle $A3<90^\circ$, the tool advances obliquely, using a whole number of rectangular movements, to the depth (L).

- 3 Machining with feed (F) in the positive direction of the longer side, in a flowing movement from inside to outside.
- 4 At the end of this process the tool is retracted from the wall and the floor in a tangent to the helix.

Finishing the side:

- 5 Rapid motion to the plunging depth (C1=).
- 6 The starting position is the first plunging depth and at least the finishing allowance (B3=) from the side. The tool moves in tangentially, mills the contour and moves away tangentially.
- 7 Repeat steps 5 to 6 until the depth (L) has been reached.
- 8 At the end of the cycle the tool moves rapidly to the 1st plus 2nd setup clearances (L1= plus L2=) and then into the centre of the pocket.

Example

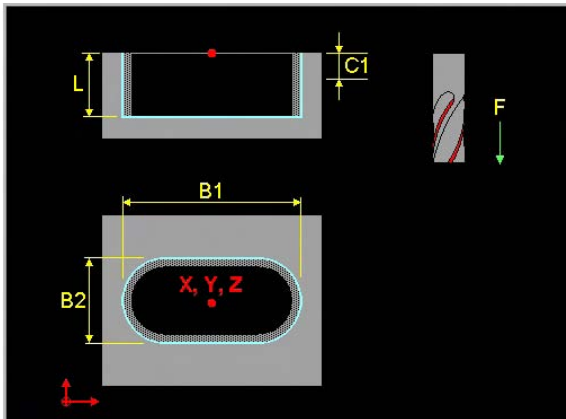


A is go obliquely to the depth. Then continuous movement.
 B is move away tangentially.
 C is move away tangentially.
 C is advance tangentially for side finishing.

Programming example	Description
N10 T1 M6 (R8 milling cutter)	Change tool
N20 S500 M3 F200	Switch on spindle
N30 G787 B1=150 B2=80 B3=1 L6 I1=1 L3=1 R20 A3=5 C2=65 C1=3	Define pocket milling roughing cycle
N40 G79 X160 Y120 Z0	Execute the roughing cycle at the programmed position
N50 G797 B1=150 B2=80 B3=1 L6 L3=1 A3=5 C1=3 C2=60 R20	Define pocket milling finishing cycle
N60 G79 X160 Y120 Z0	Execute the finishing cycle at the programmed position

11.23 G798 Key-way finishing

Define a slot milling cycle for finishing in a single program block.



```
G   Key-way finishing
B1=  1st Side length
B2=  2nd Side length
L   Depth
L1=  1st Setup clearance
L2=  2nd Setup clearance
C1=  Plunging depth
I1=  Milling 1=climb -1=conventional
F   Feed
S   Speed
```

B1= Length of the slot in the main axis.

B2= Width of the slot in the secondary axis.

The other addresses are described in the introduction to the machining cycles.

Basic settings

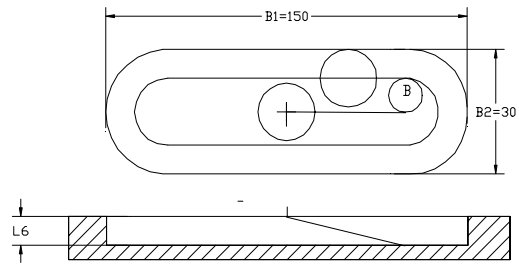
L1=1, L2=0, C1=L, I1=1

Notes and application:

Choose a milling cutter whose diameter is no greater than the width of the slot and no less than a third of the slot width.

The cycle

- 1 Rapid motion to the 1st setup clearance (L1=) above the centre of the slot.
- 2 The tool moves tangentially to the contour from the centre of the slot and finishes it in forwards rotation (I1=1).
- 3 At the end of the contour the tool moves tangentially away from the contour and floor to the centre of the slot.
- 4 The tool then moves rapidly to the 1st plus 2nd setup clearances (L1= plus L2=).

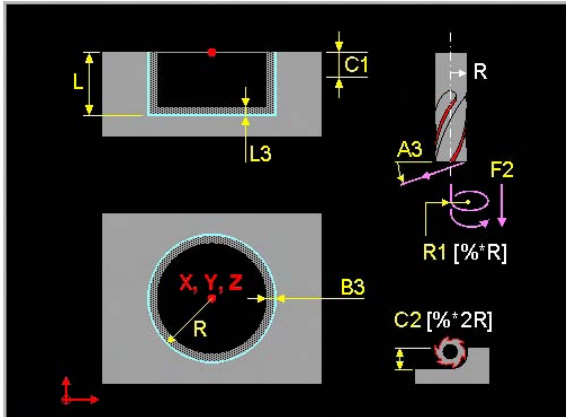
Example

B is tangential approach and retraction. Then continuous movement.

Programming example	Description
N10 T1 M6 (R8 milling cutter)	Change tool
N15 S500 M3	Switch on spindle
N20 G788 B1=150 B2=20 B3=1 L6 L1=1 A3=10 C1=3 I1=1 I2=0 F100 F2=200	Define slot milling roughing cycle parallel to the X axis
N30 G79 X20 Y20 Z0	Execute the roughing cycle at the programmed position
N40 G798 B1=150 B2=30 L6 L1=1 I1=1 F200	Define the slot milling finishing cycle, parallel to the X axis
N50 G79 X20 Y20 Z0	Execute the finishing cycle at the programmed position

11.24 G799 Circular pocket finishing

Define a circular pocket milling cycle for finishing the wall and floor of rectangular pockets in a single program block. The sides can be machined in a number of advances. This cycle allows oblique plunging into the floor and mills in a continuous spiral path.



```
G   Circular pocket finishing
R   Radius
L   Depth
L1= 1st Setup clearance
L2= 2nd Setup clearance
L3= Finishing allowance bottom
B3= Finishing allowance sides
C1= Plunging depth
C2= Proportional cutting width
R1= Proportional helix radius
A3= Plunging angle
I1= Milling 1=climb -1=conventional
I2= Finishing 0=complete 1=sides
F   Feed
S   Speed
```

```
F2= Feed for plunging
```

- B3= Allowance sides, which will be removed by finishing.
- L3= Allowance bottom, which will be removed by finishing.
- C2= Percentage of the tool diameter to be used as the cutting width on each pass. The total width is divided into equal sections.
- R1= Percentage of tool radius (>0).
- A3= Angle (0 to 90°) at which the tool can plunge into the workpiece. It only plunges vertically at 90°.
- I2= 0: Finishing wall and floor
1: Finish machining of wall only

The other addresses are described in the introduction to the machining cycles.

Basic settings

L1=1, L2=0, L3=1, B3=1, C1=L, C2=67%, R1=80%, A3=90, I1=1, I2=0, F2=0.5*F for vertical plunging and F2=F for oblique plunging.

Notes and application:

The minimum size of the pocket (R) is $2 * (\text{tool radius} + \text{finishing allowance for sides } B3=)$.

The cycle

Finishing the floor:

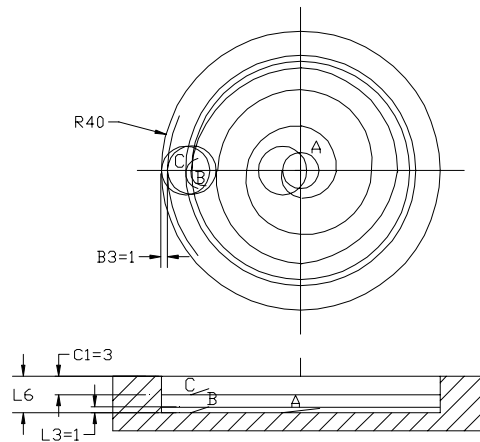
1. Rapid motion to the centre of the pocket and stay at the 1st setup clearance (L1=) above the centre of the pocket.
2. If the plunging angle $A3=90^\circ$, the tool advances with feed (F2=) to the depth (L).
If the plunging angle $A3<90^\circ$, the tool advances obliquely, using a whole number of circular movements, to the depth (L).
3. The tool then moves in a spiral path (direction depends on forward rotation (I1=1) with M3) and then clears the floor of the pocket from inside to outside.

Finishing the side:

4. Rapid motion to the plunging depth (C1=).
5. The side is then machined in a number of sections. The starting position is the first plunging depth and at least the finishing allowance (B3=) from the side. The tool then moves in tangentially, mills the contour and moves away tangentially.

6. Repeat steps 4 to 5 until the depth (L) has been reached.
7. At the end of the cycle the tool moves rapidly to the 1st plus 2nd setup clearances (L1= plus L2=) and then to the centre of the pocket.

Example



A is go obliquely to the depth. Then continuous movement over the floor
 B is move away tangentially.
 C is advance tangentially for side finishing.
 C is move away tangentially.

Programming example	Description
N10 T1 M6 (R8 milling cutter)	Change tool
N20 S500 M3	Switch on spindle
N30 G789 R40 L6 B3=1 I1=1 L1=1. L3=1 A3=5 C2=65 C1=3 F200	Define circular pocket milling roughing cycle
N40 G79 X160 Y120 Z0	Execute the roughing cycle at the programmed position
N50 G799 R40 B3=1 L6 L1=1 L3=1 A3=5 C1=3 C2=65 I1=1 F200	Define pocket milling finishing cycle
N60 G79 X160 Y120 Z0	Execute the finishing cycle at the programmed position

12. Cycles in the G800 series (Turning).

12.1 General description.

The machine and MillPlus *IT* must be prepared by the machine manufacturer for these G-functions. If not all the G functions described here are available on your machine, consult your machine handbook.

For description of these G-functions, see: chapter turning.

12.2 G822 Clearance axial.

12.3 G823 Clearance radial.

12.4 G826 Clearance axial finishing.

12.5 G827 Clearance radial finishing.

12.6 G832 Roughing axial.

12.7 G833 Roughing radial.

12.8 G836 Roughing axial finishing.

12.9 G837 Roughing radial finishing.

12.10 G842 Grooving axial.

12.11 G843 Grooving radial.

12.12 G844 Grooving axial universal.

12.13 G845 Grooving radial universal.

12.14 G846 Grooving axial finishing.

12.15 G847 Grooving radial finishing.

12.16 G848 Grooving axial universal finish.

12.17 G849 Grooving radial universal finish.

12.18 G850 Undercut (DIN 76).

12.19 G851 Undercut (DIN 509 E)..

12.20 G852 Undercut (DIN 509 F)..

12.21 G861 Threadcutting axial.

12.22 G862 Threadcutting taper.

13. Cycles in the G900 series.

13.1 General description.

The machine and MillPlus *IT* must be prepared by the machine manufacturer for these G-functions. If not all the G functions described here are available on your machine, consult your machine handbook.

For description of these G-functions, see: Manual Blum

13.2 G951 Calibration.

13.3 G953 Measure tool length.

13.4 G954 Measure length, radius.

13.5 G955 Cutter control shank.

13.6 G956 Tool breakage control.

13.7 G957 Cutter control shape.

13.8 G958 Tool setting length, radius, corner radius.

G958 TOOL SETTING LENGTH, RADIUS, CORNER RADIUS.

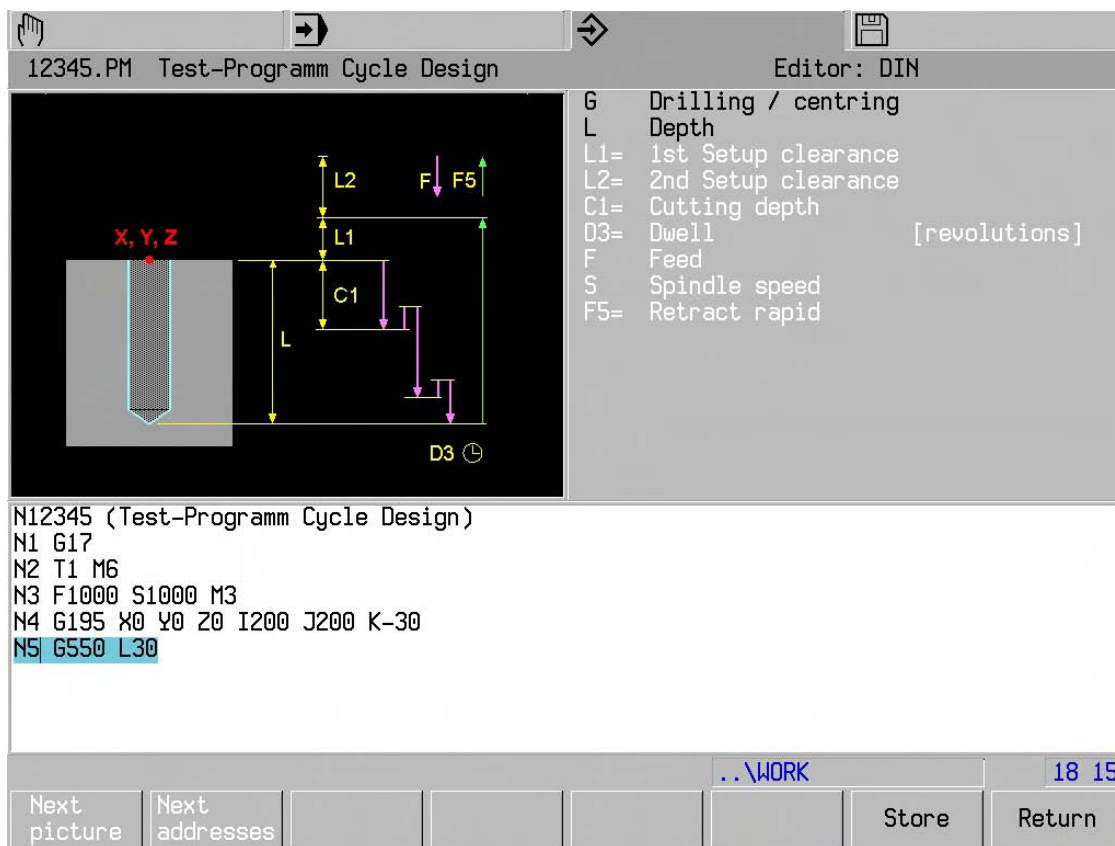
14. Cycle Design

14.1 Introduction Cycle Design

Cycle Design allows the user to define his own G functions and integrate them into the control. These G functions can be programmed within partprograms using graphics support.

The defined G function can be seen in the control in three places.

- 1) The G function list is enlarged by the new G function.
- 2) When programming, the address list is displayed with support graphics.
- 3) The G code can be seen in the partprogram.



For a G function, the following files are required:

- 1) The G function with address information and texts. File G5??.CFG (5?? is the three-digit number of the G code)
(See paragraph .2: Description of G function and addresses)
- 2) The processing macro (9995???.mm) which is called up by the G function.
(See paragraph .4: execution macro)

The following files are used as required:

- 3) Up to 10 support graphics. File *.DXF or *.BMP
(See paragraph .3: support graphics)
- 4) Up to 9 auxiliary macros (9995???.mm) which are called up by the processing macro.
(* is a serial number, one digit).
- 5) The general auxiliary macros (99905??.mm) can be called up by several processing macros. (** is a serial number, two digits).

The files for each G function are stored on the MillPlus **IT** hard disk in a dedicated directory D:\STARTUP\CYCLES\G5??.

(See paragraph .5: reading in cycle files).

Users themselves must create these directories.

Example

The files for G550 and G560 are stored in the directory D:\STARTUP\CYCLES\G550\ and ...\G560\.

Two general auxiliary macros are stored in the separate directory D:\STARTUP\.

```

D:\STARTUP\CYCLES \G550\ G550.CFG
                        < Graphic 1>.DXF
                        < Graphic 2>.BMP
                        9995500.mm
                        9995501.mm
\G560\ G560.CFG
                        < Graphic 1>.DXF
                        < Graphic 2>.DXF
                        9995600.mm
                        9995601.mm

D:\STARTUP\           1234567.mm (Maximal 7 digits)
                        *.mm
  
```

14.2 Description of G function and addresses (G5?? CFG)

Each G function and associated addresses must be described in a file (G5?? CFG).

The file defines:

- 1) The number of the G function via the file names.
- 2) The explanatory text for the G function in several languages.
- 3) For each address, a parameter list that includes: address name, E parameters, format, sign, minimum, maximum and explanatory text.
- 4) The file names of up to 10 support graphics.

14.2.1 Example- G5?? CFG file (definition G5?? CFG)

```
[DIALOG]                ; must be present.
GB                      =      Positioning
D                       =      Positionierung
NL                      =      Positionering

; Comments

[PARAMETER]
ADDRESS                =      X2
EPARAM                 =      260
OPTIONAL               =      N
FORM                   =      6. 3
DIMENSION              =      mm
SIGN                   =      Y
MIN                    =      -999999.999
MAX                    =      999999.999
GB                     =      Position
D                      =      Position
NL                     =      Positie

[PARAMETER]
ADDRESS                =      F
EPARAM                 =      252
OPTIONAL               =      Y
FORM                   =      5.0
DIMENSION              =      mm/min
SIGN                   =      N
MIN                    =      0
MAX                    =      99999
GB                     =      Feed
D                      =      Vorschub
NL                     =      Voeding

[SUPPORT]
PIC01                  =      *.DXF ;file names of first support graphic
...
PIC10                   =      *.BMP ;up to 10 support graphics
```

Remarks:

- 1) Blanks (except in texts), tabs and blank lines have no meaning.
- 3) Comments (text after ";") are permitted anywhere.
- 4) Keywords: [DIALOG], [PARAMETER] and [SUPPORT]. [PARAMETER] is to be defined for each address. The keyword [DIALOG] must be entered. Optional are [PARAMETER] and [SUPPORT]. After the keywords, information may be entered.
- 4) After [DIALOG] G function texts in several languages may be entered.
 <Language code> = For each language an explanatory text (up to 33 characters) may be entered (default setting no text)
 Only the languages required need be entered.
 The actual choice of language is determined by machine constant MC5.
 Language codes are: GB, D, NL, F, I, E, DK, S, SF, P, PL, CZ
- 5) After [PARAMETER] the address data follow.
 A maximum of 25 addresses are permitted.
 ADDRESS = Name of the address. Available address names see paragraph 2.3 (permitted addresses). Also E0 to E400 are valid names.
 All other names produce error message O141.
 If ADDRESS is missing, no address is defined.

 EPARAM = E parameter number which is assigned to the address value entered. Range 0 up to maximum number of parameters (MC 83).
 For Example: EPARAM=260 and ADDRESS=X2 means that E260 contains the programmed value of X2. If X2 is not programmed in the workpiece program, E260 is set as equal to-999999999.
 If ADDRESS=E***, the same number must be entered after EPARAM=***.
 If EPARAM is absent, no address is defined.

 DEFAULT = [Default value or MC-value with exponential factor].
 For example: DEFAULT=5 Default will be 5.
 DEFAULT=MC100,-3 Default will be contents of MC100*10⁻³.

 FORM = Determines the input format (default 6.3). 6.3 means: 6 figures before the decimal point and 3 after.
 When the address dimension [mm], [degr], [mm/min] or [diam] is, the number of digits behind the decimal point depends of MC705 and MC707.
 MC705 (Decimal digits behind the decimal point) is 3 or 4. The number or digits before and after the decimal point will be adapted.
 MC707 (Inch/Metric). is 70 (metric) or 71 (Inch). When MC707=71 the number of digits behind the decimal point will be increased by one and the number of digits before the decimal point will be decreased by one.

Overview:	Metric		Inch	
MC707	71	71	70	70
MC705	3	4	3	4
Dimensions				
[mm] Linear axis	6.3	5.4	5.4	4.5
[degr] Rotation axis	6.3	5.4	6.3	5.4
[mm/min] Feed	6.3	6.3	5.4	5.4
[diam] Diameter programming in mm	6.3	5.4	5.4	4.5

- | | | |
|-----------------|---|---|
| DIMENSION | = | Only [mm], [degr], [mm/min] and [diam] are allowed. Addresses with these dimensions are depending of MC705 and MC707.
[mm] mm for linear axis
[degr] Degree for rotation axis
[mm/min] mm pro minute for feed
[diam] Diameter programming in mm
Default: no dimension |
| SIGN | = | [Y/N] Determines whether the input can be negative or positive. (Default Y). |
| MIN | = | Smallest input (default -999999999). |
| MAX | = | Largest input (default 999999999) |
| OPTIONAL | = | [Y/N] Determinate of the input is obligatory (default is Y).
Optional Addresses are distinguished in colour (white) in the display.
If an obligatory address (not optional) is not given, the process will be stopped and an error message will be given. (from V420).
In macro must be defined a default for an optional address. |
| ACTIVE | = | [1, 2, --, 9] Stands for a coupling between graphics and addresses. The address is only shown, when the concerning graphic is chosen.
(Default: active by all graphics) (From V420) |
| <Language code> | = | For each language an explanatory text (up to 33 characters) may be entered (default setting no text)
Only the languages required need to be entered.
The actual choice of language is determined by machine constant MC5.
Language codes see [DIALOG]. |
- 6) After [**SUPPORT**] follow the file names of a maximum of 10 support graphics in *.DXF or *.BMP format. Valid file names are up to 8 characters in length, ending in *.DXF or *.BMP.
- PIC number determines the sequence of the graphics in the control.
- | | | |
|-------|---|--|
| PIC01 | = | Defines the file name of the first support graphic.
(Default no support graphic). |
| PIC02 | = | Defines the file name of the second support graphic. |
- And so on.
- 7) Changes to the G5?? CFG file will be active only after power down the machine.

14.2.2 Example-G550.CFG file

;Comments...

[DIALOG]

GB	=	Positioning
D	=	Positionierung

[PARAMETER]

ADDRESS	=	X
EPARAM	=	260
OPTIONAL	=	Y
FORM	=	6.3
DIMENSION	=	mm
SIGN	=	Y
MIN	=	-999999.999
MAX	=	999999.999
GB	=	Position
D	=	Position

[PARAMETER]

ADDRESS	=	Y
EPARAM	=	261
OPTIONAL	=	Y
FORM	=	6.3
DIMENSION	=	mm
SIGN	=	Y
MIN	=	-999999.999
MAX	=	999999.999
GB	=	Position
D	=	Position

[PARAMETER]

ADDRESS	=	E1
EPARAM	=	1
OPTIONAL	=	Y
FORM	=	6.3
DIMENSION	=	mm
SIGN	=	Y
MIN	=	-999999.999
MAX	=	999999.999
GB	=	Position
D	=	Position

[SUPPORT]

PIC01	=	ABCD1234.DXF
-------	---	--------------

14.2.3 Permitted addresses

In **CYCLE DESIGN** only the addresses below are permitted.

A	A1=	A2=	A3=	A40=	A4=	A5=	A6=	A7=	A90=	A91=	
B	B1=	B2=	B3=	B40=	B47=	B4=	B5=	B6=	B7=	B90=	B91=
C	C1=	C2=	C3=	C4=	C40=	C5=	C6=	C7=	C90=	C91=	
D	D1=	D2=	D3=	D4=							
E											
F	F1=	F2=	F3=	F4=	F5=	F6=					
G	G1=										
H											
I	I1=	I2=	I3=	I4=	I5=	I6=	I7=	I90=	I91=		
J	J1=	J2=	J90=	J91=							
K	K1=	K2=	K3=	K90=	K91=						
L	L1=	L2=	L3=	L4=	L5=	L6=	L7=				
M	M1=	M2=									
N1=	N2=										
O	O1=	O2=	O3=	O4=	O5=	O6=	O7=	O8=			
P											
Q	Q1=	Q2=	Q3=	Q4=							
R	R1=	R2=									
S	S1=	S7=									
T	T1=	T2=	T3=								
U	U1=	U2=	U7=								
V	V1=	V7=									
W	W1=	W2=	W7=								
X	X1=	X2=	X3=	X4=	X51=	X52=	X53=	X61=	X62=	X7=	X90= X91=
Y	Y1=	Y2=	Y3=	Y4=	Y51=	Y52=	Y53=	Y61=	Y62=	Y7=	Y90= Y91=
Z	Z1=	Z2=	Z3=	Z4=	Z51=	Z52=	Z53=	Z61=	Z62=	Z7=	Z90= Z91=

14.3 Support graphics

For the CNC, the graphics are to be read in a standard format (*.DXF or *.BMP).

Usually, a drawing program is used that can generate a *.DMF format from Autocad9 to Autocad14.

This *.DXF or *.BMP file must be transmitted to the control. It is then automatically converted in the control into an internal *.PIC file.

14.3.1 Making graphics in *.BMP format

When using Bitmap, the following rules must be observed:

- The dimension in PIXELS of the support graphic must be 312 by 224.
- The graphic is to be centred in the middle.
- The file size must be smaller than 64K.
- The format must be compressed. This can be done by the Microsoft Photo Editor. The Compression must be done with RLE.

14.3.2 Making graphics in *.DXF and *.PIC format

When using AutoCAD, the following rules must be observed:

- Define LIMIT [312,224]. This is the dimension in PIXELS of the support graphic.
- The graphic is to be centred in the middle.
- Define POINT [0,0] as the corner of the drawing
- Define POINT [312,224] as the second corner of the drawing
- The following are supported: LINE, POINT, CIRCLE, MARK, SOLID, TEXT, POLYLINE, LINETYPE and COLOR
- For geometry elements, the delta-X and/or delta-Y must not be greater than 255.
- A radius must not be greater than 127.
- Auxiliary drawings (ELEMENTS) must be inserted (INSERT), then perform PURGE and EXPLODE.
- Polyline must not be longer than 40 points.
- Hatching must be taken into the drawing with EXPLODE.
- Finish drawings with ZOOM and EXTEND.

In AutoCAD the next line types must be defined as under mentioned names (Only English names):



Lines ended with _2 or _3 are respectively 2 or 3 pixels width.

A *.DXF file must be generated. This *.DXF file must be converted with dxfr.exe to a *.PIC file.

Note "AutoCAD" is a registered product.

Colour code

The available colours are:

1	red	7	grey	13	pale blue
2	yellow	8	black	14	magenta
3	green	9	pale red	15	white
4	cyan	10	pale yellow	16	black
5	blue	11	pale green		
6	dark grey	12	pale cyan		

14.4 Execution macro

When executing the G function the macro with the desired sequence is started.
This macro can be written directly in DIN/ISO (*.MM).

14.4.1 Example of execution macro

```

N9995500 (positioning)
N1 (description of E parameters used)
N2 (E150 = jump parameter]
N3 (E252 = Feed]
N4 (E254 = X tool position)
N5 (E255 = Y tool position)
N6 (E256 = Z tool position)
N7 (E260 = X2_end position)
N8 (E261 = Y2_end position)
N9 (E262 = Z2_end position)
N1000 G29 I1 E150 N=1002 E150=E252=-999999999    Jump to N1002 if F is not programmed i.e.
                                                equal to -999999999
N1001 F=E252                                    activate feed F
N1002 G326 X7=254 Y7=255 Z7=256    read current position
N1003 G29 I1 E150 N=1005 E150=E260>-999999999    Jump to N1005 if X2 is programmed, i.e.
                                                greater than -999999999
N1004 E260=E254                                X2 is set equal to the current position
N1005 G29 I1 E150 N=1007 E150=E261>-999999999    Jump to N1007 if Y2 is programmed, i.e.
                                                greater than -999999999
N1006 E261=E255                                Y2 is set equal to the current position
N1007 G29 I1 E150 N=2030 E150=E262>-999999999    Jump to N2030 if Z2 is programmed, i.e.
                                                greater than 999999999
N1008 E262=E256                                Y2 is set equal to the current position
N2030 G1 X=E260 Y=E261 Z=E262    in feed to X2, Y2, Z2

```

14.4.2 Explanation

- 1) For the meanings of the G-functions see control operating instructions.
- 2) The free E parameters to be used by Cycle Design are in the range from E0 to MC83 maximum.
- 3) The programmed addresses of the cycle are transferred in E parameters from the G550.CFG file. Addresses not programmed are set equal to - 999999999. It is thus possible to recognise whether an address is programmed. This also applies to addresses such as F, which are otherwise modal.
- 4) In this example, all addresses are optional. When programming, control checks whether the value meets the criteria set from file G550.CFG. In the macro, there must be a reaction if the value is not entered. In this example, for a non-programmed Z2 address the current position is taken.
- 5) Inputs, which are mutually exclusive must be recognised in the macro. The macro may issue an error message such as P07 (value outside range).
 - 1 Changes to the *.MM file, will be only active after power down the machine.

14.5 Reading cycle files into the CNC

*.DXF, *.BMP, *.CFG and *.MM files can be loaded into the control via an Ethernet link. *.mm files can be loaded directly from a PC into the control with a suitable data communications program, such as CDS.

For example G550.cfg is to be loaded in the file directory D:\STARTUP\CYCLES\G550. Each G function has its own file directory.

Inside the CNC the text files can be produced and edited with the hard disk editor.

Changes made in the file with the editor, will be only activated after the machine switching on and off

15. Technological instructions

F-Function

To set the feed in millimetres per minute or per revolution (mm/min or mm/rev). The feedrate actually used depends on several factors, for instance material, type of machining and tool.

Format

{F..} {F1=..} {F2=..} {F3=..} {F4=..} {F5=..} {F6=..}

Definitions, abbreviations:

F: General feed for axis movements with G1/2/3
 F1=: Selection of constant cutting feedrate for radius compensation of circles.
 F2=: Retract feed at G85, infeed at G87/G89, G201 or measuring feed at G145.
 F3=: Feed for the (negative) infeed movement (infeed).
 F4=: Feed for the plane movement
 F5=: Feed unit for rotary axes
 F6=: Local feed

Type of function

Modal F, F1=, F3=, F4=, F5=
 Blockwise F2=, F6=

Notes and usage

The feed often has to be changed for technological reasons. The essential factors for making adjustments are:

- 1) Feed and direction of the movement
- 2) Constant cutting feedrate for radius compensation of circles
- 3) Feed in cycles
- 4) Feed unit for rotary axes
- 5) Local feed

15.1 F, F3=, F4= Feed and direction of the movement:

During cutting operations the feed should be carefully matched with the milling process for technological reasons. Technological conditions for milling in radial direction are different from those in axial direction.

It is most advantageous to the user if he is able to program 2 feed values modally and independently. Independent feed programming is possible with parameters F3= and F4=.

Feeds F, F3= and F4= are modal and programmable:

(0 ... 99999 [mm/min] metric)

(0 ... 9999.9 [inch/min] inch)

F3=: Sets the infeed

F4=: Sets the plane feed

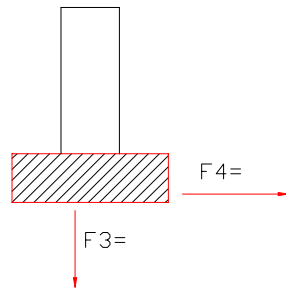
F: Sets the infeed and plane feed

If F, F3= and F4= are programmed in a block, F3= and F4= have a higher priority than F.

Tool axis: Axis perpendicular to the machining plane (G17, G18, ...).

Radial milling direction: Milling in the machining plane

Axial milling direction: Milling into the direction of the tool axis (in infeed direction only)



F4= Radial milling direction

F3= Axial milling direction

Infeed: only effective in the blocks dealing with infeed movements only.

Plane feed: effective for all other movements not involving pure infeed movements.

Initialisation: F3=0, F4=0 and F = 0

After M30, CANCEL PROGRAM Softkey or CLEAR CONTROL Softkey, F, F3= and F4= are zeroed.

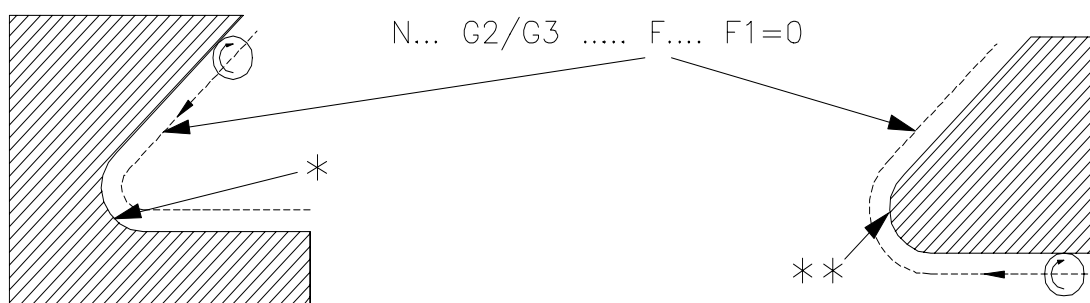
Maximum feed

The maximum feed with which the machine tool may be operated is stated in the machine documents (MC740).

15.2 F1= Constant cutting feed by radius compensation of circles

The parameter 'F1=' is used to ensure that the programmed feedrate along a workpiece contour remains constant regardless of the radius of the mill and the contour shape. This controlled velocity is called the CONSTANT CUTTING FEED.

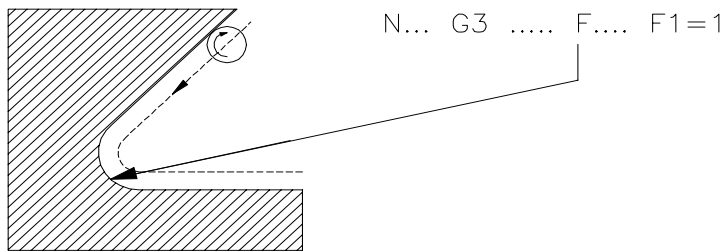
F1=0 CONSTANT CUTTING FEED not applied (default mode; also set at CLEAR CONTROL or M30 or Softkey CANCEL PROGRAM). The programmed feedrate should be the velocity of the tooltip.



* Cutting feed too high

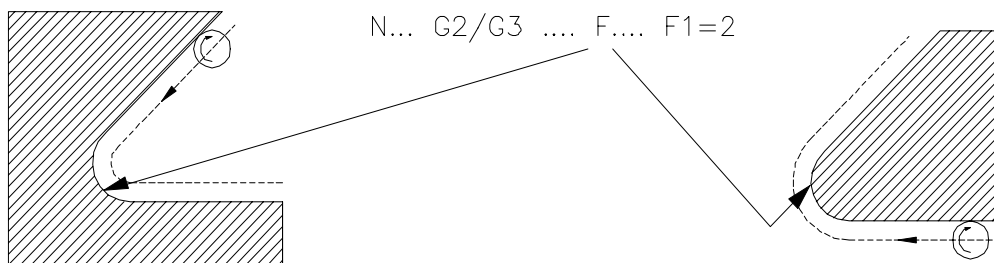
** Cutting feed too low

F1=1 CONSTANT CUTTING FEED applied only on the inside of arcs. The programmed feedrate is reduced to assure that the tooltip moves with the reduced velocity on the inside of an arc.

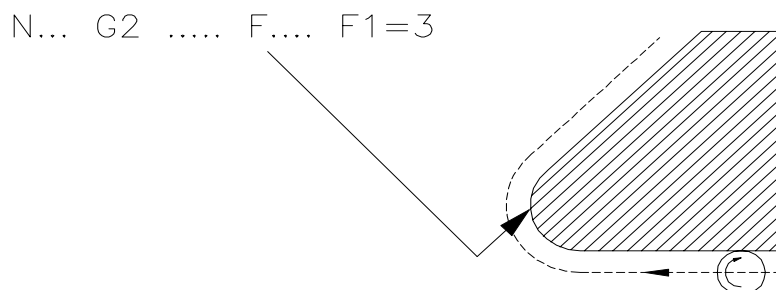


NF

F1=2 CONSTANT CUTTING FEED applied on the inside and outside of arcs. The programmed feedrate is reduced (inside arc) or increased (outside arc) to assure that the tooltip moves with the recalculated velocity. If the increased velocity is greater than the maximum feedrate (a Machine Constant value) the maximum feedrate is used.



F1=3 C.C.F. applied only on the outside of arcs. The programmed feedrate is increased to assure that the tooltip moves with the increased velocity on the outside of an area. If the increased velocity is greater than the maximum feedrate (a Machine Constant value) the maximum feedrate is used.



15.3 F2=, F3=, F4= Feed in cycles

In cycles G81, G83, G85 and G86 the movement in "axial" direction is not an infeed movement, but a feed movement. It is therefore programmed with F/F4=, not with infeed F3=.

In cycles G87, G88 and G89 the infeed movement can be programmed block-by-block using F2= and modally with F3=.

F3= is used as infeed in EASYoperate cycles.

Messages

If feeds are missing (e.g. at F3=0 or F4=0 or F0).

Message: P04 No feed is programmed

15.4 F5= Feed unit for rotary axes

G94 F5= feed of the rotary axes
F5=0 degrees/min (default)
F5=1 mm/min or inches/min)

Machines with kinematic model

Function G94 F5= is only possible if a kinematics model is defined for the machine. (MC312 must be active).

Rotary axis radius calculation G94 F5=1

In machines with the kinematics model, the rotation axis radius between the centre point of the rotary axis and workpiece can be calculated. Because of this, A40=, B40= and C40= no longer need to be programmed. The new possibility is programmed via G94 F5=1.

Shut down G94 F5=1

G94 F5=1 is cancelled by G94 F5=0, G95, programming with A40=, B40= or C40= in G0 or G1, M30, <abort program> or <reset CNC>.

15.5 F6= Local feed

F6= is a local feed which is only active in the record in which it is programmed.

F is the normal feed and also applies to the following records.

Rapid movement

A F6= in a G0 block limited the feed. The movement will be a rapid movement. That means for example that the programming logic stays active.

Example

N10 F1000

Set feed to 1000 mm/min

15.6 H Auxiliary function

Only the machine builder may use the H-function. Refer to the machine documentation.

Format

H...

Type of function

Depending on machine adaptation component.

Notes and usage

Maximum number of decimal places

The H-function can have a maximum of four decimal places.

15.7 S-function

To set the speed in revolutions per minute (rev/min) of the main spindle (S) or second spindle (S1=) of the machine tool.

Format

{S....} {S1=...}

Notes and usage

Maximum value

The maximum spindle speed of the first spindle is set in MC2491.

The maximum spindle speed of the second spindle is set in MC2691

Direction of spindle rotation

Refer to the description of M3/M4 for programming the direction of spindle rotation.

Spindle speed ranges

Refer to the description of M41/M42/M43/M44 for selecting a spindle speed range.

Technology tables

If the partprogram is entered or updated via the control, it is possible to retrieve the spindle speed from the technology tables stored in the control.

The material, type of operation and tool has to be entered too to make the selection in these tables.

Refer to the User manual for details about using the technology tables.

Example

N10 S1000

Set the spindle speed 1000 rev/min.

16. M functions

16.1 M0/M1 Program stop

M0 To interrupt the execution of a program.

M1 To interrupt the execution of a program, if this function is encountered and the Softkey OPTIONAL STOP in MACHINING is activated.

Format

{Programmed tool movement} M0 or M1

Notes and usage

Activation

The M0/M1 function will become active when the current tool movement programmed in the same block has been executed.

Spindle speed and coolant supply

The machine tool interface determines whether the spindle rotation and coolant are suppressed or cancelled as well.

Suppressed means that the spindle starts rotating and the coolant supply is switched on after resuming the execution of the partprogram.

Cancelled means that a spindle stop is executed and the coolant supply switched off. These functions have to be programmed again after the stop command.

Resume program execution

The execution of the program is resumed after the START button has been pressed.

Note: If a M1, optional stop, is programmed and optional stop mode is not active then the following G1 blocks are executed with inpos instead of inpos disregarding the G28 function. In order to get correct behaviour the programmer should program a machine function in the block next to the M1-block.

Example

N200 G1 X100 Y100 F200 M0	Move the tool to the programmed position and then halt the execution of the program.
N200 G1 X100 Y100 F200 M1	Move tool to programmed position and then halt program execution, if the Softkey OPTIONAL STOP is active.

16.2 M3/M4/M5 Spindle-rotating clockwise/counter clockwise or spindle stop

To switch on spindle rotation in clockwise (CW) or counter clockwise (CCW) direction.

M3 Spindle rotation in clockwise direction.

M4 Spindle rotation in counter clockwise direction.

To stop spindle rotation. It depends on the interface of the machine tool if the coolant supply is switched off as well.

M5 Spindle stop.

Format

{Programmed tool movement} M3 or M4 or M5

Notes and usage

Type of function

Modal

Start spindle rotation (M3/M4)

Spindle starts rotating before the tool movement programmed in the same block has been executed. The spindle starts rotating, only when the spindle speed (S) is programmed.

Cancellation

The direction of spindle rotation remains active until cancelled by:

- The opposite direction of rotation
- A SPINDLE STOP (M5 or M19)
- By END OF PROGRAM (M30) or CLEAR CONTROL.

Activation (M5)

The M5 function will become active when the current tool movement programmed in the same block has been executed.

The function remains active until a spindle rotation command is programmed.

Program stop (M0/M1) or tool change (M6/M66)

The machine tool interface determines whether the spindle rotation is suppressed or cancelled with a PROGRAM STOP or a TOOL CHANGE.

Suppressed means that the spindle starts rotating after resuming program execution.

Cancelled means that a spindle stop is executed and the direction of rotation has to be programmed again after a stop or tool change command.

Example

N20 G1 X100 Y100 S1000 M3

Start spindle rotating in a clockwise direction at 1000 rev/min before starting tool movement to programmed position.

N35 G1 X50 Y50 F250 M5

Execute tool movement and then switch off spindle rotation.

16.3 M6 Automatic tool change

To interrupt program execution and perform an automatically tool change. The execution depends of the IPLC program.

Format

{T...} {T1=...} {T2=...} M6

T	Tool identification number.
T1=	Activate/disable the cutting force monitor
T2=	Use the extra tool offsets

Notes and usage

Associated functions

M66, M67

Start tool change

The tool change is executed, before the tool movement programmed in the same block has been executed.

Sequence tool change

The execution depends of the IPLC program. Refer to the machine manual

In the following description the most commonly used sequence for the tool change command is given. Refer to the machine tool builder's documentation to see:

- if a movement to a tool change position is executed by the interface,
- Which axes are involved.
- in which order the axes will move,
- if the spindle is stopped in an oriented position,
- if spindle speed and coolant supply are suppressed or cancelled.

Machine tool with an automatic tool changer

The M6 function causes the following sequence to be performed:

- the tool first moves at rapid traverse rate to a tool change position.
- the old tool is then exchanged for a new tool and the new tools offsets made active.

Resume program execution after the automatic tool change

The execution of the program continues automatically with the movement, if any, in the block with the tool change command.

Machine tool without an automatic tool changer

The M6 function causes the following sequence to be performed:

- the tool first moves at rapid traverse rate to a tool change position.
- the execution of the partprogram is halted, to allow the user to manually change the tool.

Resume program execution after a manual tool change

After the tool has been changed, pressing the START button restarts the partprogram. The movement, if any, in the block with the tool change command is executed.

Searched tool (T)

If no T-word is programmed in a M6 block, the tool belonging to the last programmed tool number is loaded and its dimensions activated.

This situation occurs when the tool is searched during the execution of program blocks.

Spindle speed and coolant supply

The machine tool interface determines whether the spindle rotation and coolant are suppressed or cancelled as well.

Suppressed means that the spindle starts rotating and the coolant supply is switched on after resuming the execution of the partprogram.

Cancelled means that a spindle stop is executed and the coolant supply switched off. These functions have to be programmed again after the tool change command.

Tool change position

It is advised to program all axes involved with the movement to a tool change position in the block with the tool change command or in the next block. In this way MANUAL BLOCK SEARCH and RESTART after program INTERRUPT is always executed in the same way.

Incremental programming after a tool change

Increments with incremental programming are related to the last programmed position. A tool change position is not considered as a programmed position.

Example

N100: T12 M6

Interrupt program execution to allow a new tool to be loaded. Tool no.12 offsets are made active.

16.4 M7/M8/M9/M13/M14 Switch on/off coolant supply nr 2 / nr. 1

M7	Nr. 2 coolant supply	(internal coolant supply	
M8	Nr. 1 coolant supply		
M9	To switch off simultaneously the two coolant supplies.		
M13	Nr. 1 coolant supply and spindle in clockwise direction		M13=M3+M8
M14	Nr. 1 coolant supply and spindle in counter clockwise direction		M14=M4+M8

Format

{Programmed tool movement} M7/M8/M9/M13/M14

Notes and usage

Activation (M7/M8)

The coolant supply is switched on before the tool movement programmed in the same block, has been executed.

Activation (M9)

The M9 function will become active when the current tool movement programmed in the same block has been executed.

The function remains active until one of the coolant functions (M7/M8, M13/M14) is activated again.

Cancellation

The function is active until cancelled by:

- COOLANT OFF (M9)
- END OF PROGRAM (M30) or CLEAR CONTROL.

Spindle stop (M5)

It depends on the IPLC interface of the machine tool if the coolant supply is switched off with a SPINDLE STOP command.

Program stop (M0/M1) or tool change (M6/M66)

The machine tool interface determines whether the coolant supply is suppressed or cancelled with a PROGRAM STOP or a TOOL CHANGE.

Suppressed means that the coolant supply is switched on again after resuming program execution.

Cancelled means that the coolant supply is switched off and has to be programmed again after a stop or tool change command.

Machine constant for activating M13/M14

A Machine Constant (MC78) in the control must be set and the interface of the machine tool must be able to handle the functions M13 and M14. Refer to the machine tool builder's documentation to see if these functions can be programmed.

Example

N90: G1 X10 Y10 F200 M7	Switch on No.2 coolant before executing programmed tool movement.
N110: G1 X30 Y35 F150 M9	Switch off coolant supplies after executing programmed tool movement.
N120 G1 X50 Y50 F100 S500 M13	Switch on no.1 coolant supply and rotate spindle clockwise at 500 rev/min before executing the programmed tool movement.

16.5 M19 Oriented spindle stop

To stop the spindle in a programmed angular position.

See also G303 M19 D.. I2=..

Format

{D...} M19

D Angular position

Notes and usage

Associated functions

M3, M4, M5, M13, M14, M41, M42, M43, M44

Angular position (D)

The angular position is measured from a fixed position (MC2414).

Speed and Direction of rotation

Moving the spindle to the desired position always occurs in a fixed direction defined by a Machine Constant. (MC2412).

- + Speed in the positive direction of rotation (M3 or CW).
- Speed in the negative direction of rotation (M4).

Note: The D-word is only available if a Machine Constant is set. (MC2591)

Activation

The M19 function will become active, when all movements, programmed in the same block has been executed.

The spindle remains in its position, until M3, M4, M13, M14, M41, M42, M43, M44 is programmed or a M19.

Example

N125: D30 M19

Stop the spindle +30° from the fixed angular position.

16.6 M25 Measuring tool sizes

To measure tool sizes, using a measuring probe with cube-shaped probe tip.

The G50 function is used to change the stored tool sizes, if the recorded sizes are beyond the specified range of limit values.

Format

G45 [I / J / K] X1=... M25

I	+/-	2.3	Measuring direction for X axis
J	+/-	2.3	Measuring direction for Y axis
K	+/-	2.3	Measuring direction for Z axis
L	+/-	2.3	Measuring direction for B axis
X1=	+/-	6.3 5.4	Pre-measurement distance (mm or inch)

Notes and usage

G45 measuring cycle

A measuring probe with cube-shaped probe tip is used for tool measurement. The probe is mounted in a fixed position.

The measuring position is loaded to the CNC memory.

Machine constant memory

The following is stored in this memory:

- the coordinates of the fixed position of the measuring probe
- The sizes of the cube-shaped measuring probe tip.

Example

N90 G45 -I X1=5 M25

Measure the tool in negative direction of the X-axis. The pre-measurement distance is 5 mm.

16.7 M26 Calibration the measuring probe

The measuring probe radius is established by probing a calibration ring (ring gauge whose diameter is exactly known).

Format

Using the outer face of the ring gauge:

G46 [I+1,J+1 / J+1,K+1 / I+1,K+1] T... X1=... F... M26

Using the inner face of the ring gauge:

G46 [I-1,J-1 / J-1,K-1 / I-1,K-1] T... X1=... F... M26

I and J - gauge in XY plane; J and K - gauge in YZ plane; I and K - gauge in XZ plane

I	+/-	1	Measuring direction for X axis (J or K should also be indicated)
J	+/-	1	Measuring direction for Y axis (I or K should also be indicated)
K	+/-	1	Measuring direction for Z axis (I or J should also be indicated)
F	+/-	4.3 3.4	Feedrate (mm/min or inch/min)
T		7.2	Tool number
X1=	+/-	6.3 5.4	Pre-measurement distance (mm or inch)

Notes and usage

Measuring cycle

The G46/M26 measurement cycle is similar to the G46 measurement cycle. The difference between the measured centre point and the centre point stored in the machine constant memory is calculated. The F50 function is able to use this value for zero point shifts.

Machine constant memory -(MC242 / MC292 etc.).

The following is stored in this memory:

- The coordinates of the fixed position of the ring gauge
- The ring gauge diameter.

If X1= has not been programmed in the M26 block, a fixed preset value (machine constant) is used.

Example

N190 G46 I-1 K-1 T15 F50 M26

Calibrate the probe by moving it to the outer face of the ring gauge in the XZ plane. The probe radius is stored in tool memory location T15. A fixed preset value (machine constant) is used for X1=.

16.8 M24/M27/M28 Switch on/off a measuring probe

Before measurements can be performed with a remote signalling probe, e.g. an infrared probe, or a hard wired probe, the probe must be switched on and after using the probe, it must be switched off.

M24 switch on the measuring probe (Measuring probe stands on the table)

M27 switch on the measuring probe (Measuring probe is mounted in the spindle)

M28 switch off the measuring probe

Format

Switch on a probe

M24

or

M27

Switch off a probe

M28

Notes and usage

Associated functions

G145, Measuring tool G600-G609

Activation

The function M27 is executed before the movements programmed in the same block, are executed and M28 after the movements in the block.

Cancellation

The measuring probe is switched off with the function M28, the function M30 (end of program) or at CLEAR CONTROL.

16.9 M30 End of partprogram

To terminate the execution of the partprogram and jump back to begin of the program.

Format

M30

Notes and usage

Spindle rotation and coolant supply

When a M30 block is executed, the spindle is stopped and coolant supply switched off by the control.

Default settings

From G-functions belonging to one group, the default function of that group, if any, is automatically activated when the M30 function is executed.

Other functions with a default setting are reset too.

Example

N9001	Partprogram identification block.
N1 ...	Partprogram instructions.
:	
N... M30	End of partprogram and jump back to begin of program

16.10 M41/M42/M43/M44 Select spindle speed range

To select the gear range for the required spindle speed. In general the gear range is automatically selected by the control. In exceptional cases it may be necessary to change the speed without changing the gear range. This is done using the functions M41 - M44.

M41	first speed range	(MC4425)
M42	second speed range	(MC4445)
M43	third speed range	(MC4465)
M44	fourth speed range	(MC4485)

Format

S... M41/M42/M43/M44

Notes and usage

Speed range selection

The speed range can be selected automatically by the CNC; (the corresponding M-function is produced automatically by the CNC.) or by programming the corresponding M-function; useful when overlapping speed ranges are used.

Speed range limits

The limits of the spindle speed ranges are stored in the MC-memory of the CNC.

Type of speed ranges

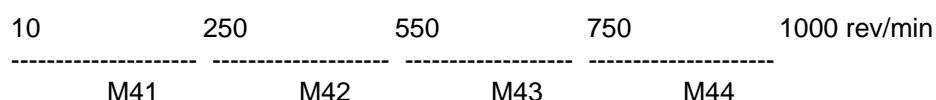
The spindle speeds can either be in separate speed ranges or in ranges, which overlap each other.

If the M-function for range selection is not programmed and a programmed spindle speed occurs in two ranges, the highest range is automatically selected.

Example

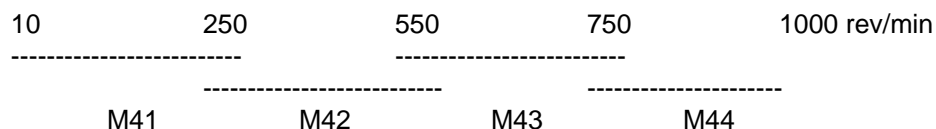
Example of speed ranges which do not overlap.

M41: 10 - 250 rev/min;
M42: 250 - 550 rev/min;
M43: 550 - 750 rev/min;
M44: 750 - 1000 rev/min.



Example of speed ranges which overlap.

M41: 10 - 250 rev/min;
M42: 200 - 550 rev/min;
M43: 500 - 750 rev/min;
M44: 700 - 1000 rev/min.



Programming example

N10 S50 M41

Assumed are the speed ranges given above. A spindle speed of 50 rev/min is required. Therefore M41 is programmed because spindle range 1 has to be used. Automatic range selection is not being used.

16.11 M66 Manuel tool change

Purpose

To interrupt program execution, to allow a manual tool change to be performed. The IPLC program decides the execution.

Format

T... {T1=...} {T2=...} M66

T	Tool identification number.
T1=	Activate/disable the cutting force monitor
T2=	Use the extra tool offsets

Notes and usage

Associated functions

M6, M67

Application of M66

The function M66 is used with a tool, which is not in the tool magazine.

Start tool change

The tool change is executed, before the tool movement programmed in the same block has been executed.

Machine tool with automatic tool changer

The M66 function is used, when a tool is required which is not in the tool magazine. The IPLC program decides the execution.

Before performing manual tool change it might be necessary to unload the spindle (by programming T0 M6) and put the tool back in the tool magazine.

It might also be necessary to program a retract to a position where the tool can be loaded.

Machine tool without automatic tool changer

The M66 function causes a halt in the program execution, to allow the tool to be manually changed. The IPLC program decides the execution.

Resume program execution

After the tool change the program is restarted by pressing the START button. The movement, if any, in the block with the tool change command is executed.

Spindle speed and coolant supply

The machine tool interface determines whether the spindle rotation and coolant are suppressed or cancelled as well.

Suppressed means that the spindle starts rotating and the coolant supply is switched on after resuming the execution of the part program.

Cancelled means that a spindle stop is executed and the coolant supply switched off. These functions have to be programmed again after the tool change command.

Example

N200 T24 M66

Interrupt program execution and change the tool manually.
The tool dimensions of T24 become active.

16.12 M67 Change tool values

To activate tool values without a change of the physical tool being performed.

Format

T... {T2=...} M67

T Tool identification number.
T2= Use the extra tool values.

Notes and usage

Associated functions

M6, M66

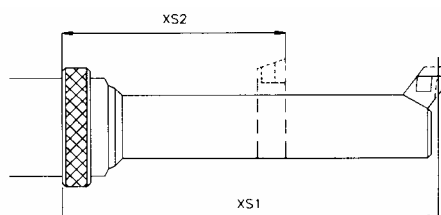
Tools with more than one cutting edge

When a tool with more than one cutting edge is used, e.g. a boring bar, each cutting edge has its specific length and radius which are stored in the tool memory as offsets and extra tool offsets for the same tool.

Activate tool dimensions

The new tool dimensions are activated, before the tool movement programmed in the same block has been executed.

Example



The above boring bar, identified as tool T12, has two cutting edges. Edge 1 with tool length XS1 is stored in the tool memory with $L=XS1$ and edge 2 with tool length XS2 with $L1=XS2$.

N100 T12 M6

N110 - N140

N150 T12 T2=1 M67

Load the boring bar; the values from T12 are used.

Use cutting edge defined with T12

A change of the offset values from XS1 to XS2. The boring bar itself is not changed.

17. T-function tool number and tool memory

- a. Description of the T-function
- b. Description of the tool memory

17.1 T-function for tool change

To enter the tool identification number in the CNC Tool Memory.
 To indicate the tool to be loaded.
 To initiate the offset values during the execution of the program.
 To initiate a search for another tool in the tool magazine.

Format

Automatic tool change
 {T...} {T2=1/2} {T1=...} M6

Manual tool change
 T... {T2=1/2} {T1=...} M66

Change offset values
 T... {T2=1/2} M67

Tool change with sufficient tool life
 T... T3=.. M6/M66

T	Tool identification number
T1=	Activate/disable the cutting force monitor
T2=	Use the extra tool offsets
T3=	Search for a tool with sufficient tool life

Notes and usage

Associated functions
 M6, M66, M67

The tool memory

A special memory is available in the control to store the tool dimensions and other tool related parameters.

Maximum number of tools in tool memory

In machine constant (MC27) is stored, the maximum of tools, which can be stored in the tool memory of the control. The maximum is 255 sets of tool dimensions.

Maximum number of tools in tool magazine

The total number of stored tools in the tool magazine is stored as a machine constant (MC28). The maximum is 255 tools.

Tool identification number (T)

The tool number in the tool memory is used to identify the tool. It is entered with the address T and a value with eight digits before and two digits behind the decimal point. The eight digits are reserved for the tool identification number. This number is also used in the partprogram and programmed with the T-words.

The two digits behind the decimal point specify a spare tool related to the tool.
 If necessary these two digits can be used in the partprogram as well.

Extra tool offsets (**T2=**)

A tool can have extra offsets. So the tool length, the tool radius and the corner radius of a mill can have three values: L, L1 =, L2=, R, R1=, R2= and C, C1=, C2= respectively.

Which offsets are used when the tool is loaded, is indicated with the word T2= in the partprogram.

T2= not programmed: the offsets stated by L, R and C are used.

T2=1: the offsets stated by L1=, R1= and C1= are used.

T2=2: the offsets stated by L2=, R2= and C2= are used.

E.g. a program block containing the words T1234 T2=2 M6 results in tool number 1234 being loaded together with its second offsets.

Tool change commands (**M6, M66, M67**)

A physical tool change is commanded by one of the functions M6 (automatic tool change) or M66 (manual tool change) and a change of tool offsets by M67. Refer to these functions for details how to use them.

Unloading the spindle

With T0 M6 the spindle is unloaded and the tool put back at the position it originally left.

Unloading the spindle is necessary:

- before a manual tool change
- with oversized tools.

Tool search

During the execution of a program, it is possible to search for the next tool in the magazine. So the tool is ready, when it should be loaded.

If the T-word is programmed without a tool change command, the search for the next tool is activated, provided that the interface of the machine tool allows a tool search.

Spare tools

A spare tool can replace the tool after its working life has ended or the lowest power level of the tool in the cutting force monitor is exceeded.

The spare tool is a two-digit number placed behind the decimal point of the tool identification number.

Selecting a spare tool

When a program block containing the words T1 M6 is executed, the CNC searches in the tool memory for the tool T1.xx.

If the tool with the lowest spare number has not been disabled, it is selected. If it cannot be used, another spare tool is selected which has a higher spare number.

If tool identification and the spare tool number are programmed, e.g. T1.05, spare tool 05 of tool 1 is used.

17.1.1 Tool life monitoring

With machine constant (MC29) the tool life monitoring will be activated.

A working tool life is assigned to a tool. Every time the tool is used, the tool life is reduced with the cutting time. When the tool life has expired, a warning message is displayed, so that the tool can be replaced.

Tool change with sufficient tool life (T3=)

The T3= word in the partprogram indicates which replacement tool with sufficient residual tool life is used for a tool change.

A tool may have several replacement tools with different residual tool lives. When T3= is programmed in a search block or during a tool change, a tool with sufficient residual tool life is searched.

Error P117 is displayed, if no tool with sufficient residual tool life is found.

Example A program block containing the words T1234 T3=1.1 M6 causes tool No. 1234 to be changed. The residual tool life is at least 1.1 minutes.

Note: When the duration of a movement smaller is than 0,1 minute, the actual tool life time will not be increased. After a lot of these small movements in the program, the operator can change the actual tool life time (M or M1=) in tool memory. For CAD-programs, which contain a lot of these short movements, a spare tool must be programmed.

Note If the life of a replacement tool expires during the machining operation, the program run is interrupted by error message P118. The program run will be resumed when the error message has been deleted and the Start key pressed. This error message is only activated if no other tools are changed during the operating sequence of Tool selection, Tool life run, Replacement tool selection and Replacement tool life run. To avoid this fault condition, the programmer may program an empty block preceding the tool change block, or he can avoid the situation described.

Note If replacement tools outside the tool magazine range have been stored in the tool memory (location > MC28), error message P117 may be activated if a replacement tool is selected during the program run. To avoid this, the replacement tools in the tool magazine range should be stored in the tool memory. This applies, at the very least, to the replacement tools that may be used during the program run. If several replacement tools are envisaged for a particular tool in the tool memory, the tools with the lowest number of allocated replacement tools should be stored in the tool magazine range.

17.1.2 Tool breakage monitoring

With machine constant (MC32) the tool breakage monitoring will be activated.

With an external device mounted on the machine tool, the tool length is measured when the tool is loaded into the spindle and again when it is put back into the magazine. If the difference between the two measurements is greater than a tolerance value, an error message is displayed and the tool disabled.

17.1.3 Cutting force monitoring (T1=)

With machine constant (MC31) the cutting force monitoring will be activated.

With an external device mounted on the machine tool, the cutting force being applied to a tool can be monitored by constantly measuring the power consumption of the spindle drive. When a power overload condition is detected, appropriate actions will be taken to prevent the workpiece or tool from being damaged.

Note

- 1 Cutting force monitoring is usually used with heavy cutting, generally with tools of >10 mm diameter.
2. Refer to the description in the machine builder manual.

Activating the cutting force monitor (T1=)

The cutting force monitor is activated with the word T1= and a value from 1 to 9999. This value is passed on the IPLC program. The T1= word is programmed in a block containing one of the tool change functions (M6 or M66).

Disabling the cutting force monitor

The cutting force monitor is disabled by programming T1=0 or by not programming the T1= word.

17.2 Tool memory

The tool memory of the control can be used to store tool dimensions and other tool related parameters. This memory can be used in a FMS environment, but also outside such an environment.

In this section a general description of the parameters in the tool memory is given.

Refer also to the machine tool builder's documentation to see which monitoring devices are activated on your machine tool, Refer also to the user manual for entering the tool data into the memory.

Addresses in tool memory:

P, T, L, L1=, L2=, R, R1=, R2=, C, C1=, C2=, L4=, R4=, L5=, R5=, L6=, R6=, G, Q3=, Q4=, I2=, A1=, E, S, M, M1=, M2=, B, B1=, Q5=, O

P	Place of the tool in the magazine
T	Tool number
L	Length
R	Radius
C	Corner radius
L4=	Length oversize
R4=	Radius oversize
G	Graphics
Q3=	Type
Q4=	Number of cutting edges
I2=	Cutting direction
A1=	Approach angle
S	Size
E	Status
M	Initial tool life
M1=	Actual tool life
M2=	Tool life monitoring
B	Breakage tolerance
B1=	Breakage monitoring
L1=	First extra length
R1=	First extra radius
C1=	First extra corner radius
L2=	Second extra length
R2=	Second extra radius
C2=	Second extra corner radius
L5=	Wear tolerance length
R5=	Wear tolerance radius
L6=	Offset length
R6=	Offset radius
Q5=	Breakage monitoring cycle (0-9999)
O	Tool orientation (only turning)

Description of the addresses

Random access tool magazine

When a tool magazine can be filled at random, a table containing per tool its place in the magazine and the corresponding tool identification number should be stored in the tool memory of the control before the first run of the program.

At a tool change (M6) the programmed tool is picked up from the magazine and the used tool put back at the empty place of the loaded tool. The table of tool places is automatically updated by the control.

Place of tool in magazine (**P**)

The three digits P-word in the tool memory are used for indicating the place of the tool in the magazine, where P1 corresponds to place 1, P2 to place 2, etc.

The actual number of places in the magazine is stored as a Machine Constant.

Tool identification number (**T**)

The tool number in the tool memory is used to identify the tool. It is entered with the address T and a value with eight digits before and two digits behind the decimal point. The eight digits are reserved for the tool identification number.

The two digits behind the decimal point specify a spare tool related to the tool.

Spare tools

A spare tool can replace the tool after its working life has ended or the lowest power level for this tool in the cutting force monitor, if available, is exceeded.

The spare tool is a two-digit number placed behind the decimal point of the tool identification number. Therefore, a maximum of 99 spare tools can be assigned to the same tool.

Tool dimensions (**L**, **L1=**, **L2=**, **R**, **R1=**, **R2=**, **C**, **C1=**, **C2=**)

A tool can have a length (L-word) and two extra length values (L1 = and L2=), a radius (R-word) and two extra radius values (R1= and R2=) and a corner radius (C-word) and two extra values (C1= and C2=).

For activating the extra tool offsets the address T2= is used.

Length- and radius oversize (**L4=**, **R4=**)

These parameters are to define the extra oversize of the tool form. The real tool length is the length (L) plus the length oversize (L4=). The real radius of the tool is the radius (R) plus the radius oversize (R4=). The length- and radius compensation works with these length- and radius-values. When no data is entered, the values are zero.

Note The length value is used with the length compensation, the radius value with radius compensation (G41 to G44 or G141), the corner radius is used with 3D radius correction (G141).

Oversized tools (**S**)

With the S-word in the tool memory is indicated if a tool occupies one place (S0) or is oversized (S1). In the latter case the tool occupies three places in the magazine, one place in which the tool is stored and two empty places at the left and right of the tool.

After using such a tool it should be put back at the same place in the magazine.

Graphic parameter (**G**)

This parameter is used to define the tool shape. The available shapes are displayed when entering tool data into the tool memory.

The tool shape is used with the graphical simulation of a partprogram.

Tool type (Q3=)

The tool type parameter in the tool memory (Q3=) has to be the same as the Q3= parameter in the technology table.

If the material, type of operation and tool number is entered, when asking for a technology proposal, this parameter is automatically picked up from the tool memory.

Number of cutting edges (Q4=)

This parameter indicates the number of cutting edges of a mill. If a technology proposal is asked for during entering a program via the control, this parameter is picked up from the tool memory, provided that the tool number and Q3= parameter are already entered.

Cutting direction (I2=)

This parameter indicates the cutting direction (3=M3, 4=M4). When no data is entered, M3 is active

Approach angle (A1=)

This parameter indicates the maximum angle for entering in the material. When this value is 90, the mill goes down perpendicular into the material.

Enabling/disabling a tool (E)

The E-word in the tool memory indicates if the tool can be used or not.

- E-1 tool is disabled, cannot be used. This parameter is set by the control if the tool life is ended or the lowest power level of the cutting force monitor is exceeded.
- E0 tool can be used, but is not measured;
- E1 tool can be used and is measured.

Wear tolerance (length and radius) for measurement cycles (L5=, R5=)

These parameters give the maximum limits for the wear tolerances by tool measurement cycles. When by measurement the wear greater is then these tolerances an error message is given.

Tool measurement offsets for measurement cycles (L6=, R6=)

These parameters indicate the positions, where the length or the radius must be measured. L6= contains the distance between the tool length (L) and the measure position.

Tool life monitoring (M, M1=, M2=)

With the word M2= in the tool memory is indicated that the tool life of a specific tool should be monitored by the CNC.

M2=0: no tool life monitoring for the specific tool

M2=1: tool life should be monitored

The word M in the tool memory is available to assign a working life in minutes to a tool. The stored tool life ranges from 1 to 99999 minutes.

Every time the tool is used, the working tool life in the memory is reduced with the cutting time. With the word M1= in the tool memory the remaining tool life is displayed.

When the tool life has expired a warning message is displayed, so that the tool can be replaced and the E-word in the tool memory set to -1 to indicate that the tool is disabled.

When the same tool is used again, program execution is either interrupted or a spare tool loaded (depending on the machine tool configuration).

A Machine Constant (MC29) must be set to indicate that tool life monitoring for the tools is required.

Tool breakage monitor (**B, B1=**)

With an external device mounted on the machine tool, the tool length is measured when the tool is loaded into the spindle and again when it is put back into the magazine. If the difference between the two measurements is greater than a tolerance value, an error message is displayed and the tool disabled.

The tolerance value is entered into the tool memory with the B-word.

With the word B1= in the tool memory is indicated that the tool breakage of a specific tool should be monitored by the CNC.

B1=0: no tool breakage monitoring for the specific tool

B1=1: tool breakage should be monitored

Breakage monitoring cycle (**Q5=**)

By hand this address it is possible to make decision in the IPLC. For example: which breakage monitoring cycle must be used. See the machine builder handbook.

Tool orientation (**O**) (Only turning)

For turning tools it is possible to tell with the tool orientation (O) in which direction the tooltip is orientated..

Refer to paragraph "Tools for turning mode".

18. E-Parameters and arithmetic functions

18.1 E-Parameter

E-parameters are useful because they allow a more flexible use of programs: one program can be used for producing different workpieces by changing the parameter values stored in the CNC's Parameter Memory.

With the aid of macros and E-parameters a problem can be solved in general terms, e.g. measuring a round hole in three or four points. At execution time the parameters receive their actual values and so the macro is adapted to the specific requirements of the program.

Format

Parameter definition
 E..=[Value or arithmetical expression]
 Parameter allocation
 [Address]= {+/-} E...
 Parameter allocation and calculation
 [Address]=[arithmetical expression]

Notes and Usage

Cancellation

Parameter values are modal unless changed by assignment of new values in a partprogram or Input via the user's panel in the parameter memory or Input from a data carrier in the parameter memory. By a softkey a parameter value or the total table can be deleted. Parameter values are not removed by softkey <CANCEL PROGRAM> or Softkey <CLEAR CONTROL> or M30.

Number of parameters

A maximum of 400 parameter values can be stored. A Machine Constant (MC83) can be used for changing this number. For system cycles (IPLC, IPP and cycles) can be use until 1250.

Address

Any of the addresses available, except the address N. Address N gives error O02 (No block number)

Parameter number (E)

This number specifies where the numerical value is stored in the parameter memory.

Use of a parameter in more than one program

Different programs can use a parameter. If a new program uses a parameter, which already has been assigned a value by a previous program, a new value must be assigned to the parameter otherwise the old value will be used again.

When a parameter is programmed but no value for that parameter is present in the Parameter Memory, an error message (P28) is produced (Value not defined).

Remark:

During execution of an in IPP generated program, E5, E6, E40 till E100 will be used. In BTR-mode E0 will be used.

Standard types of parameters

Parameters can be used in every CNC PILOT system. If the option EXTENDED ARITHMETICAL OPERATIONS is not available, the type of the parameter can be:

- | | | |
|---|----------------------------|------------|
| 1 | Integer, no decimal point: | E1=20 |
| 2 | Fixed point value: | E1=200.105 |
| 3 | Floating point value: | E1=1.965e5 |

A floating-point value is comprised of a fixed-point number (the mantissa) which is multiplied by an exponential value e.g. 1.965e5 means $1.965 \cdot (10^5)$, which is equal to 196500.

Input accuracy

The input accuracy of the parameter types is:

- Integer: a 15-digit number
- Fixed point: at least 6 decimals behind the decimal point, maximum 15 decimals behind the decimal point
- Floating point: The mantissa is programmed as a fixed-point value; the exponent is an integer between -99 to +99.

Parameter table Display.

The parameters stored in the Parameter Memory can be displayed on the screen.

The displayed values are rounded values with a restricted number of decimals. They are displayed either as a fixed-point value or as a value in the so-called scientific notation, thus with an exponent.

Though the calculated stored value has the same or a greater accuracy as the displayed one, the stored value might be different.

E.g.	Stored in the memory:	99.99999999999999 (more than 16 digits)
	Displayed:	100

Arithmetical operations

The four arithmetical operations of addition (+), subtraction (-), multiplication (*) and division (:) are available in every CNC system.

E1=E2:	set E1 value equal to E2 value.
E1=E2+E3:	add the E3 value to the E2 value and store the result in E1.
E1 =E2-E3:	subtract the E3 value from the E2 value and store the result in E1
E1 =E2*E3:	multiply the E2 value by the E3 value and store the result in E1
E1=E2:E3:	divide the E2 value by the E3 value and store the result in E1.

Restrictions

1. Arithmetical expressions must not contain spaces between characters.
E.g. E1 = E2 is not allowed. This should be E1=E2
2. Arithmetical operators must be between arithmetical values e.g.
E1=E2 E3 is not allowed. Must be E1=E2*E3.
3. Consecutive arithmetical operators are also not allowed e.g.
E1=E2*:E3, except in the case of E1=E2*-E3.
Only one arithmetical operation is allowed in an expression.

18.2 Arithmetical functions

Format

Arithmetical operations

Exponentiation	$E1=E2^E3$
Square root	$E1=\text{sqrt}(E2)$
Absolute value	$E1=\text{abs}(E2)$
Integer conversion	$E1=\text{int}(E2)$
Value of Pi(=3.141592..)	pi
Whole number conversion with large value	$E1=\text{ceil}(E2)$
Whole number conversion with small value	$E1=\text{floor}(E2)$
Rounding	$E1=\text{round}(E2,n)$ (n is decimals)
Remainder of division	$E1=\text{mod}(E2,E3)$
Sign	$E1=\text{sign}(E2)$
Maximum	$E1=\text{max}(E2,E3)$
Minimum	$E1=\text{min}(E2,E3)$

Note: In all the cases, it is permitted to replace the E-Parameter between the parentheses by an arithmetical expression (up to four levels) e.g. $E1=\text{sqrt}(E2^2+E3^4)$.

Trigonometrically and inverse trigonometrically functions

Sine	$E1=\text{sin}(E2)$
Cosine	$E1=\text{cos}(E2)$
Tangent	$E1=\text{tan}(E2)$
Arc sine	$E1=\text{asin}(E2)$
Arc cosine	$E1=\text{acos}(E2)$
Arc tangent	$E1=\text{atan}(E2)$
Arc sine	$E1=\text{asin}(E2,E3)$
Arc cosine	$E1=\text{acos}(E2,E3)$
Arc tangent	$E1=\text{atan}(E2,E3)$

Relational expressions

equal	=	$E1=E2=E3$
not equal	<>	$E1=E2<>E3$ true $E1=1$
greater	>	$E1=E2>E3$
greater or equal	>=	$E1=E2>=E3$ not true $E1=0$
less	<	$E1=E2<E3$
less or equal	<=	$E1=E2<=E3$

Parentheses (...(...(...(.....)...)...)...): maximum of four levels

Notes:

1. If the relational expression is true $E1=1$, if not true $E1=0$. This can be used in the function for conditional jump (G29)
2. On a data carrier the mentioned functions should be in lower case characters.
3. In the format description, the parameters E2 and E3 represent any parameter or expression.
4. Functions and arithmetical expressions can also be used without parameters, e.g. $X=(10+12*\text{sin}(23))$.
5. The E-parameter containing the result of a calculation of a mathematical function, which has the required accuracy, but different decimals can be stored.

E.g. $E1=99.9999999$ and $E1=100.0000001$ are two values with the same accuracy, but different decimals.

Problems may arise when using the function "int" or a relational expression in which all digits are compared.

Block length

The size of an expression is restricted to 40 characters. A program block can contain a maximum of 255 characters. This restricts the number of expressions that can be in a program block.

Converting calculated values to program words

Parameter (or calculated) values are automatically 'rounded' and converted by the CNC to the fixed number of decimals belonging to the program word.

E.g. programming $E1=101.74e-3$ and $X=E1$ causes the value to be rounded, so that $X0.102$ is obtained. The value is reduced to three digits after the decimal point.

18.2.1 Arithmetical operations

Exponentiation (raising to a power)

$E1=E2^2$ or $E1=E2^{E3}$ (with $E3=2$)

Both of the above operations result in the $E1$ parameter being made equal to the square of the $E2$ value.

Exponentiation operations are performed in a fixed sequence. The exponentiation operation is performed first and then the effect of the sign is included.

For example, the equation of $E1=-3^2$ is evaluated by first performing exponentiation (3^2) and then including the effect of the sign resulting in a negative value (-9).

If a negative number has to be raised to a power, the number should be enclosed by parentheses e.g. $E1=(-3)^{E3}$. Another method is to assign the negative number to a parameter and then perform the exponentiation operation on the parameter e.g. $E2=-3$ and then $E1=E2^2$.

Not allowed exponentiation calculations are:

- i. 0^0 ;
- ii. $E2^{E3}$, when $E2 < 0$ and $E3$ has a real value.

Reciprocals

The reciprocal of $E2$ can be calculated by $E1=1:E2$ or $E1=E2^{-1}$

Quadrates

The quadrate of $E2$ can be calculated by $E1=E2 \cdot E2$ or $E1=E2^2$

Square roots

The square root of $E2$ can be calculated by:

$E1=\text{sqrt}(E2)$ or $E1=E2^{.5}$

$E1=\text{sqrt}(\dots)$: between parentheses an arithmetical expression is allowed e.g. $E1=\text{sqrt}(E2^2+E3^4)$.

Parameter $E2$ must be positive or zero when a square root (sqrt) calculation is performed.

Absolute values

When the absolute function is used a negative value becomes positive. Positive values remain unchanged.

$E1=\text{abs}(E2)$

Whole numbers

When using the integer function, the numerical value is truncated, i.e. all figures after the decimal point are ignored.

$E1 = \text{int}(E2)$

Example: $E2=8.9$ results in 8, $E2=-8.9$ results in -8

Note: The integer function is changed with the floor function in V420 and higher

Whole number with smallest integer bigger or equal to argument

When using the integer function with the smallest value, the numerical value is rounded according to the largest argument.

$E1 = \text{ceil}(E2)$

Example $E2=8.9$ results in 9, $E2=-8.9$ results in -8, $E2=8$ result in 8

Whole number with largest integer smaller or equal to argument

When using the integer function with the largest value, the numerical value is rounded according to the smallest argument.

$E1 = \text{floor}(E2)$

Example $E2=8.9$ results in 8, $E2=-8.9$ results in -9, $E2=8$ result in 8

Rounding

When the rounding function is used, the numerical value is rounded according to the number of decimal places.

$E1 = \text{round}(E2, n)$ (n is number of decimal places)

Remark If the number of decimal places is not entered, zero is assumed.

Example: $n=1$ and $E2=8.94$ results in 8.9 $n=1$ and $E2=-8.94$ results in -8.9
 $n=1$ and $E2=8.96$ results in 9.0 $n=1$ and $E2=-8.96$ results in -9.0

Remainder of dividing

When the remainder function is used, the remainder is returned by the argument.

$E1 = \text{mod}(E2, E3)$

Remarks: $E1 = E2 - \text{int}(E2:E3) * E3$

If $E3$ is 0, $E2$ is returned.

If $E3$ is not entered, 1 is assumed.

The sign is equal to the sign of $E1$.

Example $E2=5$ and $E3=3$ results in 2, $E2=-5$ and $E3=3$ results in -2

Sign

When the sign function is used, the sign is returned.

$E1 = \text{sign}(E2)$

Example $E2=8.9$ results in 1, $E2=0$ results in 0, $E2=-8.9$ results in -1

Note:

1. The E-parameter is stored with the highest accuracy, but the user should be aware that different digits could be stored.

E.g. $E1=99.9999999$ $E3=100.0000001$

$E2 = \text{int}(E1)$ gives $E2=99$ $E2 = \text{int}(E3)$ gives $E2=100$

Both parameters $E1$ and $E3$ are stored with the same accuracy, the display shows in both cases the value 100, but the result of the function "int" is different.

2. It is advised to add a small value, e.g. the required accuracy of the calculations, to the parameter of which the integer value is to be taken.

Example So if $E1=99.9999999$ or $E1=100.0000001$, the expression $E2 = \text{int}(E1 + .0000001)$ gives $E2=100$ independent of the value from $E1$.

So if $E1=-99.9999999$ or $E1=-100.0000001$, the expression $E2 = \text{int}(E1 + .0000001)$ gives $E2=-100$ independent of the value from $E1$.

Maximum

The function max() returns the maximum value of the two arguments.

$E1 = \max(E2, E3)$

Example $E1 = \max(16, -10)$ gives $E1 = 16$

Minimum

The function min() returns the minimum value of the two arguments.

$E1 = \min(E2, E3)$

Example $E1 = \min(16, -10)$ gives $E1 = -10$

The constant PI

The value of pi is stored in the control with an accuracy of 15 digits. At each place, where a value or E-parameter is allowed, the word pi can be used. This constant can be used e.g. with the conversion of angles from radians to decimal degrees or visa versa.

Angle in decimal degrees

The default programming mode for an angle is in degrees and decimal parts of a degree. This value can be entered directly in the trigonometric functions, arithmetical or relational expressions.

Example. $E1 = \sin(44.209303)$

Angle in radians

Sometimes with calculations in which angles are involved, it is useful to express the angle in radians.

360° equal $2 \cdot \pi$ radians.

Therefore an angle of 44.209303° is equal to 0.7715979 radians.

If with the trigonometric functions the angle is in radians, the word rad has to be added to the value, thus:

Example $E1 = \sin(.7715979 \text{ rad})$

Angle conversions

Degrees, minutes and seconds to decimal degrees:

An angle of $44^\circ 12' 33.5''$ is converted into a decimal equivalent as follows:

N... $E1 = 44 + 12:60 + 33.5:3600$.

This conversion produces a decimal degree value of $E1 = 44.209303$.

Decimal degrees to radians

An angle of 44.209303° is converted into radians as follows:

N... $E1 = (44.209303:360) \cdot 2 \cdot \pi$

This conversion produces an angle in radians of $E1 = .7715979$

Radians to decimal degrees

An angle of 0.771579 radians is converted into decimal degrees as follows:

N... $E1 = (0.771579 \cdot 360):(2 \cdot \pi)$.

This conversion produces a decimal degree value of 44.209303.

18.2.2 Trigonometrically and inverse trigonometrically functions

Trigonometrical functions

The following trigonometrically functions are available:

sinus (sin), cosine (cos), tangent (tan),

These are written as:

$E1=\sin(E2)$ $E1=\cos(E2)$ $E1=\tan(E2)$

For example, the sine of the angle 44.209303° can be programmed in any of the following ways:

$E1=\sin(44.209303)$ or $E1=\sin(0.7715979 \text{ rad})$

For example, the sinus (sin), cosine (cos), tangent (tan), of the angle $44^\circ 12' 33.5''$ can be programmed in any of the following ways:

$E1=\sin(44' 12'' 33.5)$

Notes:

1. Parameter E2 represents any arithmetical expression.
2. Odd multiples of 90° cannot be used with the tan function; if this occurs an error message is generated.
3. If the angle is in radians, the word rad has to be programmed with the trigonometric functions.

Inverse trigonometrical functions

The following inverse functions of the trigonometrically functions are available:

arcsin (asin), arccos (acos), arctan (atan).

These are written as:

$E1=\sin(E2)$ $E1=\cos(E2)$ $E1=\tan(E2)$

Notes:

1. Parameter E2 represents any arithmetical expression.
2. The values of the inverse functions asin and acos should be between -1 and +1; atan can have any numerical value.
3. The angle produced by these functions, is in decimal degrees.
4. The angle produced by asin and atan will be between -90° and $+90^\circ$.
5. The angle produced by acos will be between 0° and 180° .

Also possible:

$E1=\sin(E3,E4)$ $E1=\cos(E3,E4)$ $E1=\tan(E3,E4)$ where $E2=E3:E4$

Remark: - $\text{abs}(E2)$ must be less than or equal to 1 for acos and asin.
 - the angle created lies between 0° and $+360^\circ$

18.2.3 Relational expressions

Relational expressions

The purpose of a relational expression is to set an E-parameter value to 1 when some conditions are met. If these conditions are not met the value of the parameter is set to 0.

This parameter can be used to perform jumps in the program by means of the G29 function.

The following relations can be used:

equal	=	$E1=E2=E3$
not equal	<>	$E1=E2<>E3$
greater	>	$E1=E2>E3$
greater or equal	>=	$E1=E2>=E3$
less	<	$E1=E2<E3$
less or equal	<=	$E1=E2<=E3$

Example

N.. G29 E1=E2>E3 E1 N=400

This block means, If parameter E2 is greater than E3, the relation is true and thus parameter E1 set equal 1. Parameter E1 is used in the G29-block as the jump condition. So if E2>E3 a jump to N400 is executed.

Note:

1. Parameters E2 and E3 represent any arithmetical expression.
2. To satisfy a relational expression all digits are compared and have to be the same. When parameter values are produced by calculations, this may cause difficulties. In this case limits have to be set and checks performed to ensure that the value is within these limits.

Priorities of evaluating arithmetical and relational expressions

Arithmetical and relational operations are performed by the CNC in following order:

1. Evaluate functions: sin, cos, tan, asin, acos, atan, sqrt, abs, int.
2. Calculate reciprocals (\wedge^{-1}) or perform exponentiation (\wedge).
3. Multiply (*) or divide (:).
4. Add (+) or subtract (-).
5. Evaluate relational expressions (=, <, >, >=, <=).

When a block contains operations, which have the same priority, the operations are evaluated from the beginning of the block to the end.

The block E1=3+7:2-4²+5*6 is evaluated in the following order:

1. $4^2 = 16$
2. $7:2 = 3.5$
3. $5*6 = 30$
4. $3+3.5 = 6.5$
5. $6.5 - 16 = -9.5$
6. $-9.5 + 30 = 20.5$

18.2.4 Parentheses

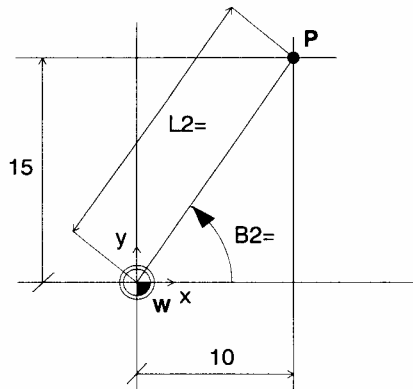
Use of parentheses ()

Parentheses () can be used to group operations and thus impose a different order of evaluating an expression. The expression between parentheses is evaluated in the normal sequence. Refer to PRIORITIES OF EVALUATING ARITHMETICAL AND RELATIONAL EXPRESSIONS for this sequence. After evaluating the expression the result is used.

A pair of parentheses can be placed within another pair; this is known as 'nesting'. The expression between each pair of parentheses is evaluated starting from the innermost 'nested' pair to the outermost pair. A maximum of four pairs of parentheses can be used in one expression.

Examples

EXAMPLE 1 Calculation of polar coordinates.



If the polar coordinates of point P related to the program datum W has to be calculated, the programming could be:

N100 B2=atan(15:10) L2=sqrt(10^2+15^2)

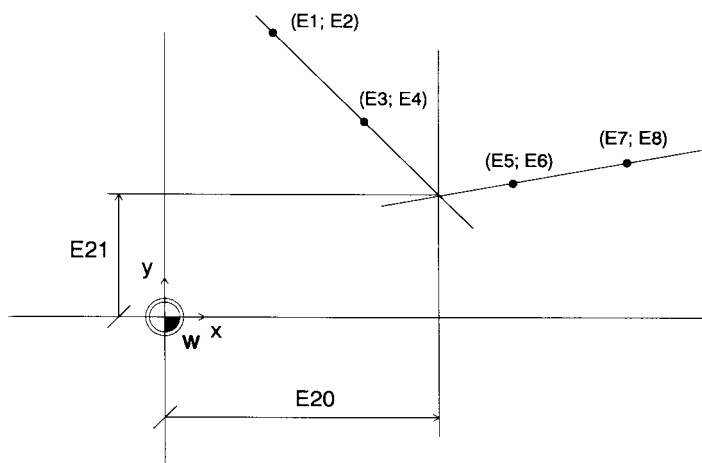
For B2= the calculation is performed in the sequence:

- Calculate 15:10
- Determine angle in decimal degrees

For L2= the calculation is performed in the sequence:

- Calculate 10^2
- Calculate 15^2
- Add 10^2 and 15^2
- Extract the square root.

EXAMPLE 2 Calculation of the intersection point of two lines.



Input parameters

- | | |
|----|--|
| E1 | first coordinate of first point on first line. |
| E2 | second coordinate of first point on first line. |
| E3 | first coordinate of second point on first line. |
| E4 | second coordinate of second point on first line. |

E5 first coordinate of first point of second line.
 E6 second coordinate of first point on second line.
 E7 first coordinate of second point on second line.
 E8 second coordinate of second point on second line.

Output parameters

E20 first coordinate of intersection point.
 E21 second coordinate of intersection point.
 E79=1 an error detected in the macro.
 =0 no error.

The macro

N99401 (CALCULATE INTERSECTION POINT TWO LINES)

N1 E11=E3-E1 E12=E4-E2 E79=0 the unit vector of the first line is calculated.
 N2 E13=sqrt(E11^2+E12^2)
 N3 E11=E11:E13 E12=E12:E13
 N4 E13=E7-E5 E14=E8-E6 the unit vector of the second line.
 N5 E15=sqrt(E13^2+E14^2)
 N6 E13=E13:E15 E14=E14:E15
 N7 E16=E11*E13+E12*E14 check to see if the unit vectors are not parallel.
 N8 G29 E15=abs(E16)<.99995 N=12 E15
 N9 E79=1 if the lines are parallel, parameter E79 is set an error displayed with a program stop. After the start the calculations are not performed.
 N10 M0 (LINES ARE PARALLEL)
 N11 G29 E79 N=17
 N12 E15=E1-E7 E16=E2-E8 compute the factor of the vector.
 N13 E17=(E15*E12-E16*E11)
 N14 E17=E17:(E13*E12-E11*E14)
 N15 E20=E7+E17*E13 calculate the coordinates of the intersection point.
 N16 E21 =E8+E17*E14
 N17

Remark Parameter E79 can be used to handle the error in the activating program or macro.

Example of how the macro is used

First line through the points (30,50) and (60,30).
 Second line through the points (100,50) and (50,10).

The calculation of the intersection point could be programmed as:

N100 E1=30 E2=50 E3=60 E4=30 the points on the first line.
 N101 E5=100 E6=50 E7=50 E8=10 the points of the second line.
 N102 G22 N =99401 calculation of the intersection point.
 N103 G29 E79 K0 N=... if an error is detected, transfer control to block, which contains M30.
 N104 G0 X=E20 Y=E21 move with rapid traverse to the intersection point.

19. Turning

19.1 Introduction

The turning mode has been developed for machines with a C axis that can turn continuously. In this way, turning operations can be carried out on a milling machine.

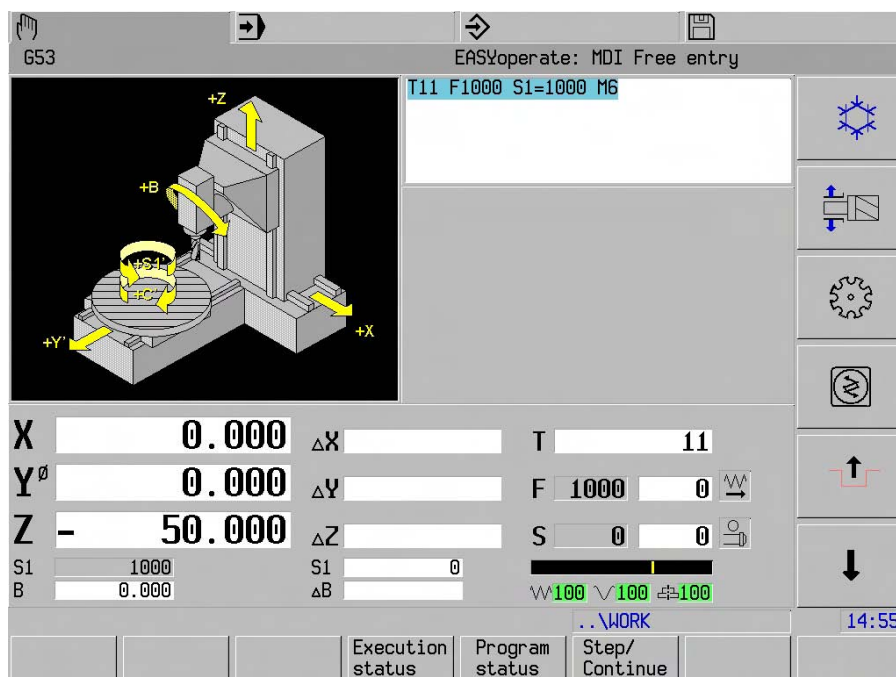
The C axis can be switched to turning mode. The C axis is then programmed as a turning spindle via S1= and M1=. The turning tools are mounted in the milling spindle and clamped at the desired orientation.

In special cases, the milling spindle can be programmed parallel to the turning spindle via S and M. A second milling spindle is not possible on machines with turning mode.

Notes and application

AVAILABILITY	Machine and CNC must be prepared for turning mode by the machine builder. If your machine is not equipped with all the G functions described here, please refer to your machine manual.
GRAPHICS	The graphic is not displayed symmetrically to the rotation.
DISPLAY	If G36 is active, the display of the C axis position changes to display S1=. S1= is the spindle revolution (G97) or constant cutting speed (G96). The axes display for the axes X and Y can optionally be changed to diameter via: manual operation mode, options and axes display. The programming remains in radius. Only when the turning mode is activated, the axes display is changed from radius to diameter The machining status is expanded with G36/G37.
REFERENCE POINT	When the controller runs up, it is always in milling mode G37. The C axis can only be switched to turning mode after the reference points have been approached.
ZERO POINT	In turning mode, the workpiece zero point in X should lie in the centre of rotation of the S1 axis. It is recommended that the workpiece zero point in Y should also lie in the centre of rotation of the S1 axis.
SPINDLE OVERRIDE	Spindle override is effective for both spindles in turning mode (G36).

Screen in turning mode



19.2 Machine constants

Machine constants for turning

Machine constants	Description
MC 268	Second Spindel (0=no, 1=yes)
MC 314	Turning mode (0=off, 1=on) Activated: - G functions G36 and G37 - Turning cycles - Machine constants MC2600 - MC27xx, MC45xx
MC 450	Balancing: measurement axis (1=X, 2=Y, 3=Z) This MC determines the axis on which the rotary table is installed. Unbalance is easiest to measure in this axis. Normally, 2 = Y axis The MC is used in the 'unbalance calibration' (installation), G691 'unbalance detection' and G692 'unbalance checking' cycles.
MC 451	Balancing: maximum amplitude [μm] This MC specifies the permissible residual amplitude in the measuring axis. The measurement is cancelled if the measured amplitude is greater than MC451 at a particular speed. Normally 5 [μm]. The MC is used in the 'unbalance calibration' (installation), G691 'unbalance detection' and G692 'unbalance checking' cycles. The C1 parameter can be superimposed on this in the G691 and G692 cycles
MC 452	Balancing: initial radial position [μm] This MC specifies the radial position (distance from centre point) of the rotary table (S1 axis) at which a balancing mass is normally mounted to compensate for unbalance. The MC is used in the G691 'unbalance detection' cycle.
MC 453	Balancing: rotary table displacement [mGrad] This MC specifies the 0 position of the rotary table and the position (door) where the operator fits the mass to compensate (and calibrate) the unbalance. The MC is used in the 'unbalance calibration' (installation) and G691 'unbalance detection' cycles.
MC2600 - MC2799, MC4500 - MC4599	Second spindle

19.3 G36/G37 Switching turning mode on and off

- G36 Switches the machine from milling mode on the C axis to turning mode with turning spindle S1.
 G37 Terminates turning mode. Switches the machine back to milling mode

Format

N... G36 or N... G36

Parameters

none.

Type of function

modal

Notes and application

G36

The CNC switches the C axis to turning mode.

In turning mode, the circular axis is programmed as a second spindle using S1= and M1=. C parameters can no longer be programmed.

The display of C (setpoint and actual value) on the screen is switched to S1. If the turning spindle is stationary, the position (0-359.999 degrees) is displayed.

G95 is active, assigned to the second spindle.

All G functions can be programmed, but not all the G functions are meaningful. For instance, a pocket has no meaning in turning mode. The C parameters and certain other parameters can no longer be programmed in certain G functions.

A survey of permitted G-Functions can be found in section 14

The effect of G36 remains active until it is cancelled by G37, runup or <CNC reset>. G36 is not cancelled by M30 or <Cancel program>.

G37

The CNC switches the C axis on again.

If the rotary spindle is still turning at the start of G37, it is first stopped.

The position of the circular axis is displayed on the screen with a value between 0 and 359.999 degrees.

G94 becomes active.

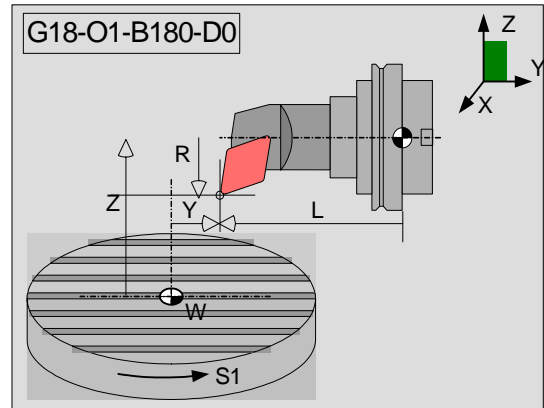
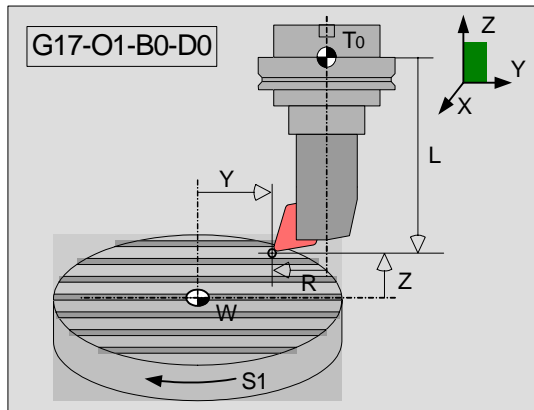
The effect of G37 remains active until it is cancelled by G36. G37 is not cancelled by M30 or <Cancel program>. G27 is always active following runup or <CNC reset>.

Program example	Description
N9000 (C-Axes operation)	
N1 T.. M06	ActivateTurning tool
N2 G0 Y.. Z..	Tool positioning
N3 G74 X1=1 Y1=1	Rapid movement to table center
N4 G54 I1	Zero point table center X0, Y0
N5 G36	Activate turning mode
N6 G17 Y1=1 Z1=2	Activate working plane
N7 G96 M1=3 S1=200	Constant cutting speed and spindel direction
N8 G302 O7	Tool orientation override
N9 G..	Turning machining
N10 G37	Switch-off turning mode
N11 G..	Milling machining
N12 M30	Program end

19.4 G17/G18: Machining planes for turning mode

In the turning mode the machine tool can machine work pieces in the different machining planes. The machining plane is defined in the turning mode (G36), with:

- G17 Y1= 1 Z1=2, tool axis Z (vertical) or
- G18 Y1= 1 Z1=2, tool axis Y (horizontal)



The function G17/G18 defines, in which axes (Y/Z) the tool corrections for length (L) and radius (R) are calculated:

- G17: L in Z-direction, R in Y-direction
- G18: L in Y- direction, R in Z- direction

In the turning mode machining can be performed in both the YZ or XZ- machining surface as individual DIN-commands. With the machining cycles however, machining can be performed only in the YZ- machining surface.

Remark:

- Y1=1 (first main axis); Z1=2 (second main axis)
- The angle (positive) and circular direction (CW) are defined from the Y-axis to the Z-axis.
- The G37 switches the actual G17/G18-plane in the turning mode back to its G17/G18-plane in the milling mode.
- The tool radius (R) is calculated in the different G17/G18-planes as a shift. Depending of the tool orientation (O) the compensation is calculated in the relevant Y or Z-axis.

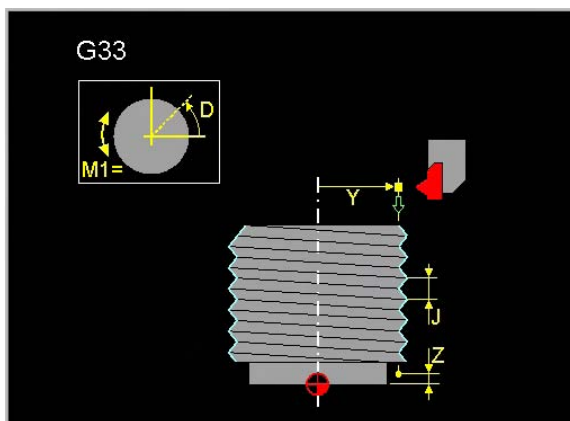
19.5 G33 Thread cutting

G33 is a thread-cutting movement. In a single pass it cuts a thread with feed and fixed pitch. The feed is determined by the spindle speed and the pitch.

Characteristics:

- Thread cutting is carried out with an open positioning control loop. Possible thread types: cylindrical and conical
- Spindle and feed override are ineffective during G33
- A number of thread movements can be programmed in sequence (e.g. oblique entry and exit)
- The lead angle of the thread can be programmed.
- The speed (S1=) and direction of rotation (M1=) must be pre-programmed

G33 is signalled to the IPLC (WIX thread movement)



```
G   Single threadcutting movement
X   Endpoint coordinate
Y   Endpoint coordinate
Z   Endpoint coordinate
J   Pitch
D   Start angle threadcutting
?90= Endpoint abs. (X,Y,Z..)
?91= Endpoint incr. (X,Y,Z..)
```

Notes and application

USE

G33 movement commences:

- when the actual and programmed spindle speeds are equal (actual N=target N) and
- after the marker and the calculated lead angle D

G33 carries out a single thread cutting movement from the current position to the programmed point.

The programmed speed (G97 S1=) and lead (J) determine the axial feed rate.

G33 stops at the end of the movement with an accurate stop and G1 is modally active.

Notes: If the pitch or speed is not programmed, there is no G33 movement; the axis remains stationary:

- if the pitch J or speed S1= is not programmed, an error message (P02/P26) is issued
- the direction of spindle rotation M1= 3 or 4 has no effect on the direction of movement
- Speed and Feed override are not effective during G33 movement and are switched to 100%

INTERRUPTION

It is possible to interrupt thread cutting by:

- stopping the feed: Movement stops at the end of a G33 movement.
- stopping the feed/spindle: Spindle and movement stop at the end of a G33 movement.

Notes: If a number of G33 movements are programmed in sequence, the machine stops after the last G33 movement.

MACHINING PLANE

G33 can only be executed within one turning plane

MODES

- G33 is inoperative in MDI mode: Error code P77.
- In single block operation a number of G33 movements are executed in sequence.

TEST RUN / GRAPHICS

In graphics and in the test run without MST, G33 runs like G1.

PROGRAMMING EXAMPLE

Programming example	Description
N9000 (thread cutting)	
N1 T.. M06	Change thread cutting tool
N1 G0 Y.. Z..	Position the tool
N2 G36	Switch on turning mode.
N3 G17 Y1=1 Z1=2	Activate machining plane
N4 G97 M1=3 S1=100	Speed and direction
N7 G0 Y.. Z..	Advance to starting position
N8 G0 Y..	Adjust to cutting depth
N9 G33 J2 Z91=..	Thread cutting to end point
N10 G0 Y..	Retract
N11 G0 Z..	Return to starting position
N7 G37	Switch on milling mode
N6 M30	Program end

19.6 G94/G95 Expanded choice of feed unit

Informs the CNC how to evaluate the programmed speed (S).
 This function is expanded for turning mode.
 The spindle and the circular table must be programmed for turning.

Notes and application

In addition, the rotary table (second spindle) must be programmed with S1= and M1= for turning.

In milling mode (G37): N... G95 F.. {S..} {M..}

In turning mode (G36): N... G95 F.. {S1=..} {M1=..}

S and M refer to the spindle

S1= and M1= refer to the second spindle

PRIORITY

The active spindle speed is either S or S1=. If S and S1= are both programmed, S1 is used.

MAXIMUM SPEED

The value of the second spindle speed (S1=) lies between 0 and 'Max. output voltage speed' (MC2691).

MACHINE FUNCTION

Second spindle machine functions:

- M1=3 second spindle clockwise
- M1=4 second spindle anticlockwise
- M1=5 second spindle stop

Positioning of the second spindle (M1=19) is not possible. Positioning takes place in milling mode.

The S1= and M1= addresses can also be programmed in the following G functions: G0, G1, G2, G3, G94.

The G95 function calculates the feed in [mm/min (inches/min)] based on the programmed feed in [mm/rev], [inches/rev] and the active spindle speed.

19.7 G96/G97 Constant cutting speed

G96 Programming constant cutting speed.
G97 Switching off constant cutting speed..

Format

N... G96 F.. D.. {S..} {M..} {S1=..} {M1=..}
N... G97 F.. {S..} {M..} {S1=..} {M1=..}

Parameters

G Constant cutting speed
D Upper speed limit (rev/min)
F Feed
S Cutting speed (m(feet)/min)
M Machine function
S1= Cutting speed (m(feet)/min)
M1= Machine function

G Spindle speed
S Speed (rev/min)
M Machine function
S1= Speed (rev/min)
M1= Machine function

G96

S and M refer to the spindle
S1= and M1= refer to the second spindle (rotary table)

G97

Type of function

modal

Notes and application

MAXIMUM SPEED (D)

The value of the second spindle speed lies between 0 and 'Max. output voltage speed' (MC2691).

MACHINE FUNCTION

Second spindle machine functions:

- M1=3 second spindle clockwise
- M1=4 second spindle anticlockwise
- M1=5 second spindle stop

Positioning of the second spindle (M1=19) is not possible. Positioning takes place in milling mode.

The G96 function calculates the feed in [mm/min (inches/min)] based on the programmed feed in [mm/rev], [inches/rev] and the active spindle speed.

The active spindle speed is either S or S1=. If S and S1= are both programmed, S1 is used.

19.8 Turning tools in the tool table

Tool data

The most relevant tool data that is stored in the the tool table for turning tools is listed below:

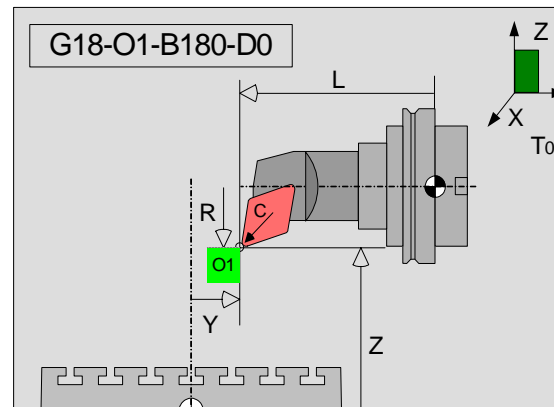
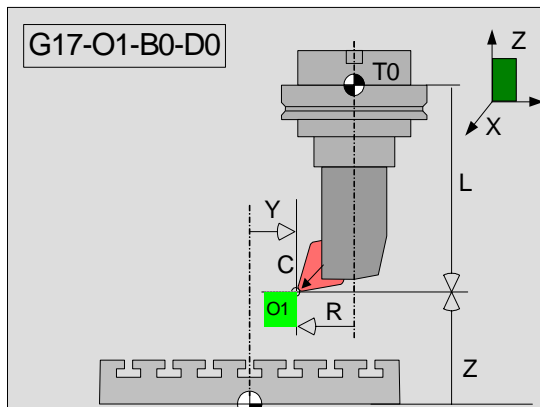
L	Length
R	Radius
C	Corner radius
Q3	Tool type
G	Graphics
O	Orientation

Tool correction

The tool dimensions are stored in the tool table as tool length L and tool radius R. How these dimensions are calculated in the relevant axes, depends on the actual plane (G17/G18) and tool nose position (orientation O):

- G17: Tool length L in the Z-Axis; tool radius R in the Y-Axis
- G18: Tool length L in the Y-Axis; tool radius R in the Z-Axis

The radius (R) is considered to be a shift and is calculated, depending on the tool orientation (O) with sign (+/-) in the relevant axis.



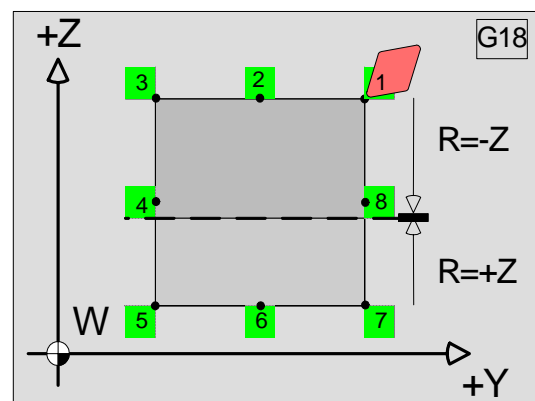
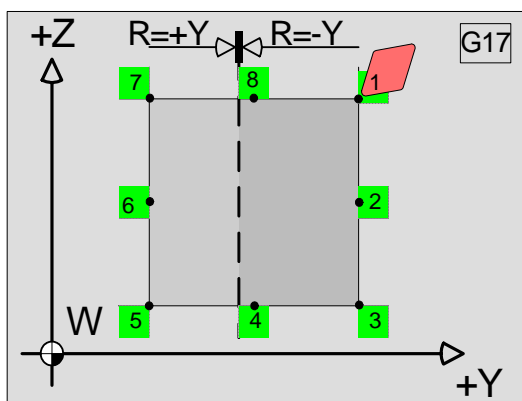
Tool orientation (O)

The tool orientation (O) determines in which direction the tool nose cutting edge is positioned. It calculates and compensates the tool path in the respective axes with two parameters for:

- Tool radius (R)
- Tool nose radius (C)

Tool radius compensation (R)

The pictures below show in which axis the tool radius in the G17/G18-plane is calculated.



The table below shows the relation between G17/G18, R, C and the way the radius is calculated.

	Plane	Orientation	Radius correction	Radius as shift
G17	G17	Not active	R	Not active
	G17 Y1=1 Z1=2	1, 2, 3, 4, 8	C and O	R in negative Y-direction
		5, 6, 7	C and O	R in positive Y- direction
G18	G18	Not active	R	Not active
	G18 Y1=1 Z1=2	1, 2, 3, 4, 8	C and O	R in negative Z- direction
		5, 6, 7	C and O	R in positive Z- direction

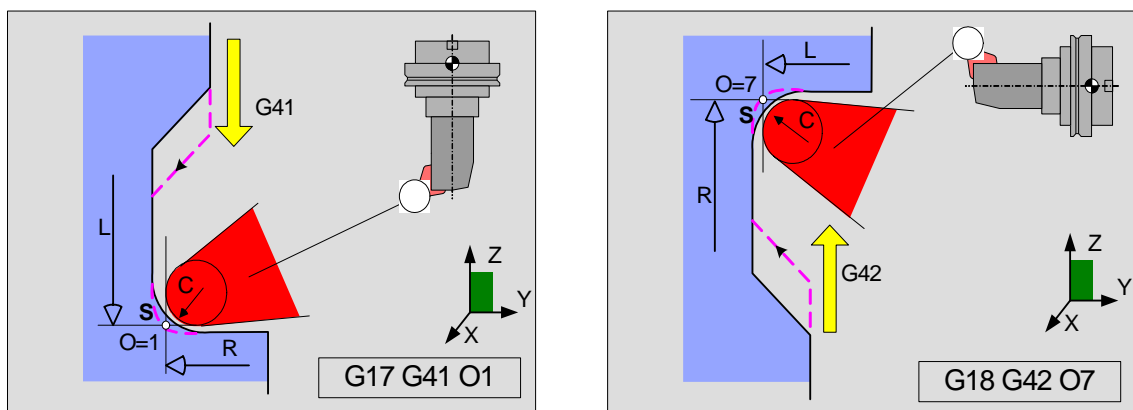
Remark:

- Tool nose radius compensation refers to the tool tip corner radius C.
- Radius compensation refers to the tool radius R.
- The tool orientation O is taken from the tool table but can be overwritten by the G-function (G302 Ox) in the program.

Tool nose radius compensation (TNR)

Turning tools have a nose radius (C) on the cutting edge. During machining of e.g. conicals, phases and radii, inaccuracy problems occur which can be corrected by the tool nose radius compensation **TNR**.

Programmed movements are related to theoretical tool cutting point (S). Contour errors appear at contours that are not axes parallel. The **TNR** calculates a compensated tool path, **equidistant**, to correct this error.



The pictures above show a turning tool in the different machining planes G17/G18.

The turning tool is performing a single cut with G1/G3 and is situated:

- At the left side of the contour (G41) with orientation O1 (picture on the left) and
- At the right side of the contour (G42) with orientation O7 (picture on the right)

Considered is the tool nose. The tool nose tip, with its radius (C), is considered to be as a circular plate, whereby its backside is able to cut the contour. The clearance angle of the tool (back side of the plate) must be appropriate to prevent the contour from damage during cutting.

Tool nose radius correction (TNR) switching on/off

The **TNR** is calculated at all clearance- and grooving cycles.

At DIN-programming (G1/G2/G3) the **TNR** can be switched on/off additionally. The **TNR** is switched on/off with the following G-functions:

- G40: **TNR** is switched off
- G41: **TNR-on**, the turning tool is on the left from the contour side
- G42: **TNR-on**, the turning tool is on the right from the contour side

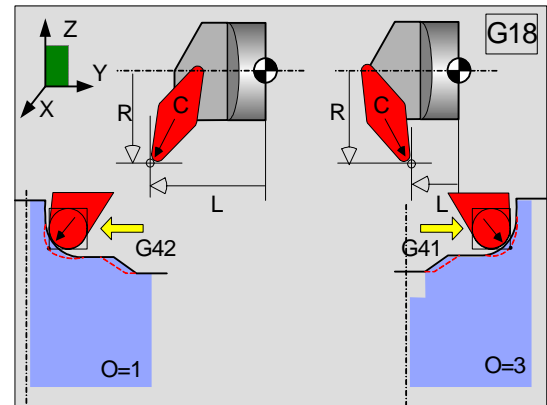
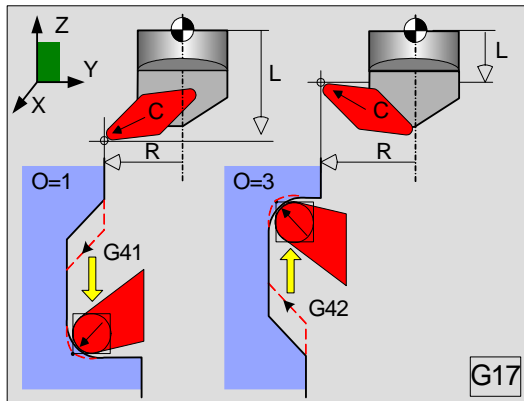
Examples of TNR in G41 and G42

In the pictures below two examples are shown of a turning application.

- The left picture shows a turning application in the axial axis in G17:
 - G41 and O1 (Left side)
 - G42 and O3 (Right side)
- The right picture shows a turning application in the radial axis in G18:
 - G42 and O1 (Left side)
 - G41 and O3 (Right side)

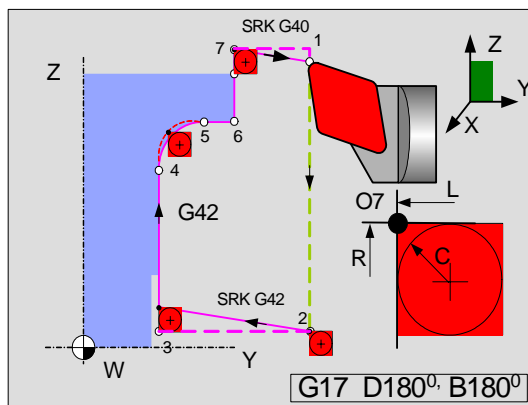
Note in the pictures:

- The swivel head position
- The different cutting edges

**TNR Start/Stop**

The picture below shows, as an example of the DIN-program N171842.PM, the way **TNR** is switched on and -off.

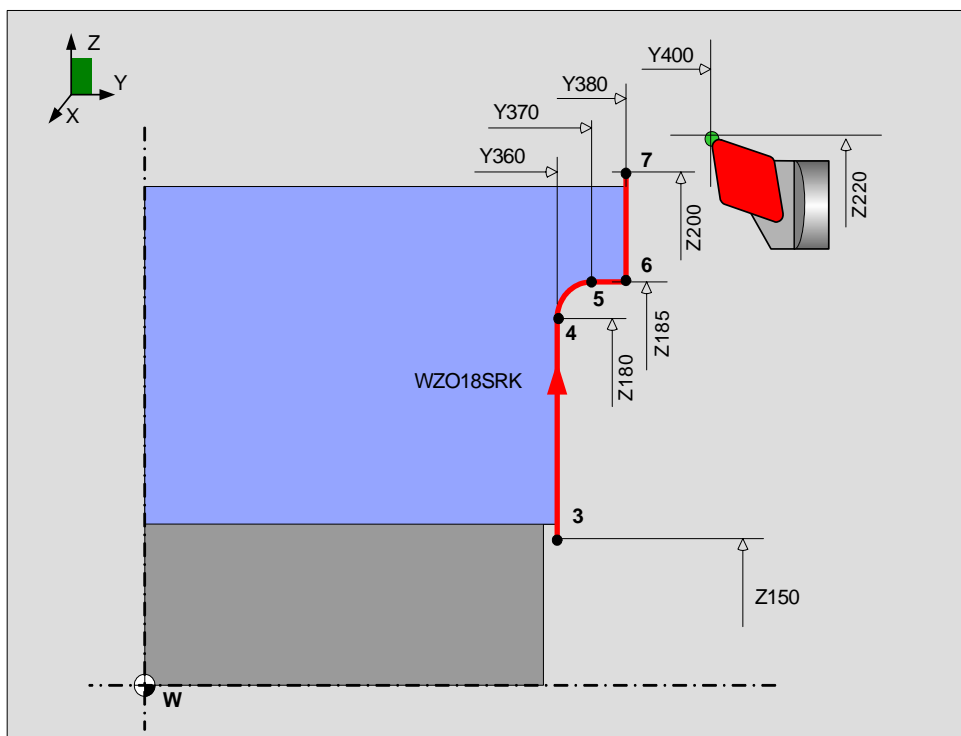
- Note:**
- The tool must have enough lead and trail cut at switching on and -off **TNR** in order to cut the complete contour.
 - Switching on and -off **TNR** must be programmed perpendicular to the contour side



Example DIN-Program

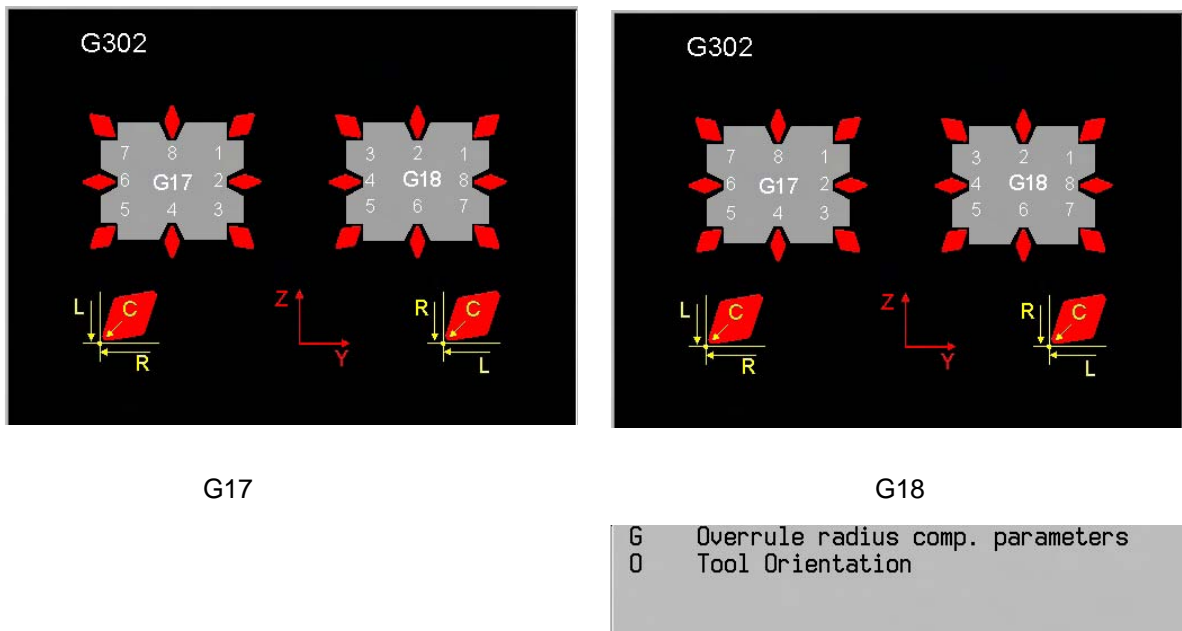
Program example	Description
N171842 (Contour cutting)	
N1 G195 X0 Y0 Z0 I0 J300 K300	Graphic window definition
N2 G54 I10	Zero point shift to table centre
N3 G0 X0 Y450:2 Z250	Tool displacement
N4 T10 M06	Tool exchange turning tool
N5 G36	Turning mode active
N6 G17 Y1=1 Z1=2	Machining plane G17 active
N7 B180	Tool head swivel
N8 G0 Y400:2 Z220	(1) Tool positioning
N9 G96 M1=3 S1=200 D500	Constant cutting speed and table direction
N10 G302 O7	Tool orientation O7
N11 M52	Main spindle release
N12 M19 D0	Tool orientation
N13 M51	Main spindle clamp
N14 G0 Z150	(1→2) Positioning
N15 G42	TNR switching on G42
N16 G1 Y360:2	(2→3) Contour side approach with G42
N17 G1 Z180	(3→4) Contour side cutting
N18 G2 Z185 Y370:2 R5	(4→5) Radius cutting
N19 G1 Y380:2	(5→6) Contour side cutting
N20 G1 Z200	(6→7) Contour side cutting
N21 G40 Y400:2 Z220	(7→1) Positioning with G40 (TNR switching off)
N22 G97 S1=100	Turning table in G97-mode
N23 G37	Milling-mode active
N24 M30	Program end

Work piece drawing



19.9 G302 Override radius comp. parameters

The G302 function determines the tool orientation during execution. The tool parameters in the tool memory are not changed.



- O Defines the tool orientation used during execution.
The value lies between 0 and 8.

Type of function

Non-modal

Notes and application

Remarks:

If the active tool orientation is overwritten, the direction of the R displacement may also change.

In G18, the active tool orientation is already changed by the CNC. See chapter 'Tool correction'.

USE

The G302 function should be used if, for example, the main spindle has been turned through 180 degrees with M19 D90. In this case, the orientation is mirrored compared with the status with M19 D90. The orientation should also be mirrored when turning takes place 'across the centre'.

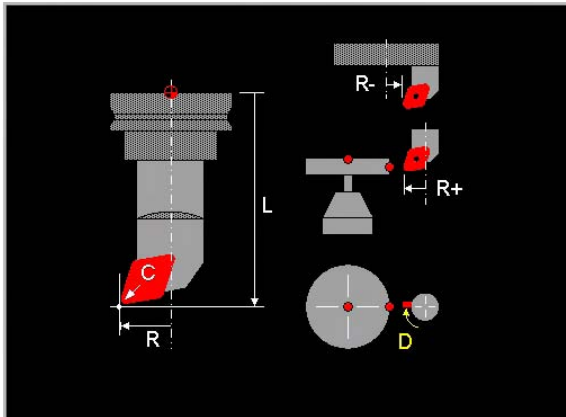
Note: In these cases, the direction of rotation of the 2nd spindle should also be reversed.

DELETING

G302 is switched off again with G302 without parameter, set plane (G17, G18, G19), tool change, M30 and <Cancel program>

19.10 G611 TT130: Measure turning tools

This cycle measures the length and radius of turning tools. Only tools in the G17 machining plane are measured.



```
G  TT130: Turning tool measurement
D  Orientation angle tool tip
I1= Clearance
I4= Measuring: 0=L+R 1=L 2=R
```

Notes and application

INPUT PARAMETERS

D

The tool

tip must always be located in the correct position before measuring, i.e. with its tip parallel to the axis and perpendicular to the measuring device. Since the turning tool can be at any angle during machining, depending on the type of work, the operator decides whether the tool measuring position (D) is programmed into the measuring cycle.

I1= Safety distance (I1=)

The safety distance in the direction of the spindle axis must be sufficient to prevent any collision with the workpiece or clamping devices. The safety distance is with respect to the top edge of the stylus. Basic setting (I1=0)

I4= Measuring: 0=L+R 1=L 2=R (as desired)

The tool length and radius are measured as standard

Notes: Both the position and direction of the tool are reset after measuring.

- If the angle of orientation is not known (no spindle reference run) error message P339 is issued.

- If neither the orientation nor the position of the tool is known, error message P334 is issued.

- Only tool orientations (O1 and O7) are allowed for measurement with TT-120. If a different tool orientation is given, error message R326 (tool orientation not allowed) is issued.

TOOL PARAMETERS FROM THE TOOL TABLE

The measuring cycle uses the following parameters from the tool table.

Parameters	Description
L*	Tool length
R*	Tool radius
C	Cutting radius of tool
L4=	Length allowance
R4=	Radius allowance
L5=	Length tolerance
R5=	Radius tolerance
E	Tool status
O	Tool orientation

Important: Make sure that the length (L) and radius (R) entered are within the tolerance (MC397), otherwise there will be an error message.

- Note:** - Before measuring the tool for the first time, enter the estimated radius, the estimated length and the tool orientation of the tool concerned in the tool table.
 - The measuring cycle adopts the current O from the tool table or from G302

THE CYCLE

MillPlus **IT** measures the tool in accordance with a fixed programmed sequence:

1. The machining plane for measurement is set
2. The tool axis moves to the safety distance (I1=)
3. The current tool position is checked and reset if it is not correct for measurement
4. Both axes advance to the measuring position of the probe
5. The tool axis advances to the probe
6. The tool length is measured first, followed by the radius
7. The tool axis moves up to the safety distance
8. The R/L measured values (first measurement) or the tolerance R4=/L4= (check measurement) are saved
9. The original working plane, tool position and tool orientation are reset

MEASURE TOOL (E=0 or no value)

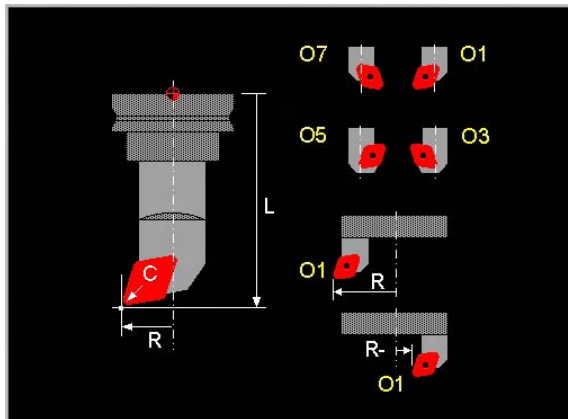
On the first measurement MillPlus **IT** overwrites the tool radius R and the tool length (L) in the tool memory and sets the allowance R4 and L4=0.

CHECK TOOL (E=1)

If you are checking a tool, the measured tool data is compared with the data in the tool table. MillPlus **IT** calculates the deviations with the correct sign and enters these in the tool table as allowances R4 and L4. If one of the dimensions is greater than the allowable wear (L5= and R5=) or breaking tolerance an error message is issued.

19.11 G615 laser system: L/R measurement of turning tools

This cycle measures the length and radius of turning tools. The turning tool is measured when stationary in both the G17 and G18 planes. Only turning tools with tool orientation 1 or 7 can be measured.



```
G  Laser: Turning tool measurement
D  Orientation angle tool tip
O  Tool orientation
```

Notes and application

INPUT PARAMETERS

- D** Tool position for measuring position
In the safety position, the tool is oriented to the programmed position (D). The tool tip must then be parallel to the axis and at right angles to the laser.
- O** Tool orientation
The orientation (O) of the tool tip determines whether measurement takes place in front of the laser or behind it. Only values 1 or 7 are allowed.

TOOL PARAMETERS FROM THE TOOL TABLE

Parameters	Description
L	Tool length
R	Tool radius
C	Cutting radius of tool
L4=	Length allowance
R4=	Radius allowance
L5=	Length tolerance
R5=	Radius tolerance
L6=	Length measurement offset
R6=	Radius measurement offset
E	Tool status
O*	Tool orientation

Note: -The tool length (L) and radius(R) must be entered accurate to +/- 5mm
 - The tool cutting radius (C) should preferably be entered
 - The orientation O is not used in the measuring cycle

TOOL TYPES

Turning and plunging tools can be measured with the main and secondary cutter to the rear (see illustrations on the right)

LENGTH AND RADIUS MEASUREMENT

The tool length (L) and radius (R) must be stored in the tool memory. Before the first measurement the approximate length and radius must be entered (max. deviation +/-5mm).

Note: incorrect input can lead to error messages or even collision with the laser light cabinet.

CORNER RADIUS

We recommend always entering a corner radius (C) in the tool memory. The cycle then runs faster.

ACTIONS

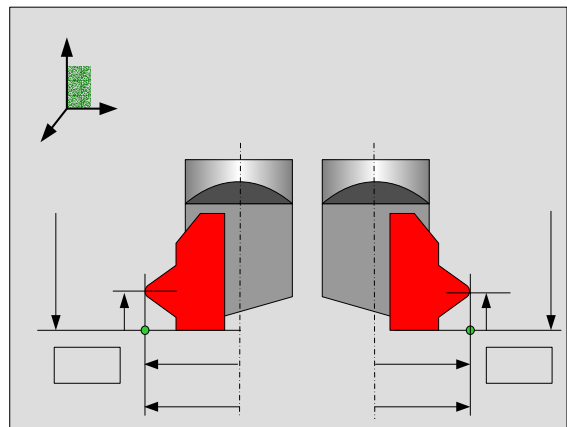
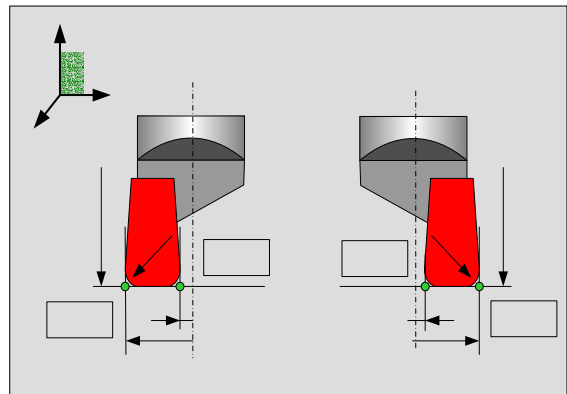
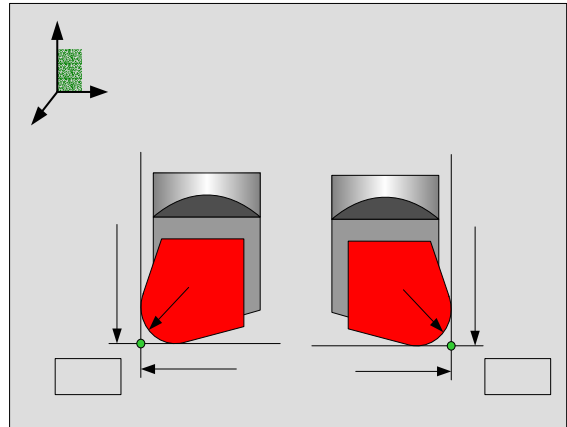
- Measure tool (E=0 or no value)
On the first measurement the tool length (L) and radius R are overwritten, the allowance $L4=0$ / $R4=0$ and the tool status $E=1$ are set. If a corner radius C is entered, this is also corrected.
- Check tool (E=1)
The measured deviation is added to $L4=$ / $R4=$ in the tool table

THE CYCLE

1. At the start of the cycle the axes move rapidly to the safety position using positioning logic.
2. In the safety position, the tool is orientated to the programmed position (D) and clamped there.
3. The tool moves into the measuring position at measuring speed.
4. The measurement is carried out.
5. After the measuring process the Z axis moves back to the safe position

Notes: The cycle can be called in milling mode and in turning mode.

- The tool can be measured both in front of and behind the laser. The greatest accuracy is reached when the tool is measured in the machining position. After completing the cycle, the spindle remains in the programmed position (D) and the orientation before measurement (O) is active.



19.12 Unbalance cycles

19.12.1 General information

To machine workpieces to be turned on an FP machine, both the machine (rotary table) and workpiece must be balanced, otherwise the life of the machine, the quality of the workpiece or even the safety of the operator cannot be guaranteed.

First, the unbalance properties of the rotary table must be determined. Usually, this unbalance calibration takes place when the machine is handed over or during servicing.

To determine the unbalance of the clamped workpiece, a new cycle has been introduced: **G691 unbalance detection**.

This cycle can be called up directly in manual mode under the FST menu.

The result is a suggestion for compensating for the measured unbalance: what mass should be attached at what radial position from the turning centre. The rotary table is automatically turned to the position where the mass should be attached.

The radial position for an available compensating mass can be calculated in the dialog window. The relationship between mass and position are shown graphically.

To ensure that no turning operations take place in automatic mode with too great an unbalance, a new G function can be called in the program: **G692 unbalance check**.

This G function checks the unbalance present against the permissible unbalance. If this is exceeded, an error message is issued, following which the operator can cancel the automatic mode and carry out a new unbalance detection with correction in manual mode

19.12.2 Description of unbalance

When working in turning mode, centrifugal forces occur if the clamped part (e.g. a pump housing) has an unbalance. This influences concentric accuracy because the second spindle (= circular axis C) is configured on the Y axis.

Unbalance $U = m \cdot R$

where:

m = mass [g]
 R = distance from centre of mass to centre of table [mm]

The unbalance is given in [gmm] (grammes*mm). This means that 500 [grammes] at 300 [mm] (=150000 [gmm]) has the same effect as 1000 [grammes] at 150 [mm].

The centrifugal force is proportional to the unbalance and rises quadratically with rising speed.

Centrifugal force $F_c = m \cdot R : 1000000 \cdot (S : 2 \cdot \pi : 60)^2$

where:

F_c = centrifugal force [N]
 m = mass [g]
 R = distance from centre of mass to centre of table [mm]
 S = speed [rpm]

The unbalance must be compensated by a balance weight. For this purpose, the available measuring systems of the circular axis C and the linear axis Y are used to detect the unbalance that exists.

19.12.3 (G227/G228) Unbalance monitor

This function monitors the unbalance that occurs during machining when a part that has not been balanced is being turned on a milling lathe. If a defined limit is exceeded machining stops. There are two such limits, one fixed limit that can be set and one programmable limit. The fixed limit is set by the machine manufacturer and is always active. It is set 'higher' with the purpose of protecting the machine. The programmable limit is 'lower' and is switched on as required, for example not during feed movements.

Note:

- The current unbalance value is displayed in the 'Spindle performance display'.
- The unbalance monitor function can be switched on and off in the program.

SWITCHING ON THE UNBALANCE MONITOR (G228 I1=, I2=, I3=)

I1= Defined when the MillPlus **IT** generates an error message n28 'Unbalance monitor 1: Excessive unbalance'

- | | | |
|-----|-----------------|-----------------------------------|
| 0 = | Feed movement: | no error message (Basic setting). |
| | Rapid movement: | direct error message |
| 1 = | Feed movement: | error message at end of contour |
| | Rapid movement: | direct error message |
| 2 = | Feed movement: | error message at end of block |
| | Rapid movement: | error message at end of block |
| 3 = | Feed movement: | direct error message |
| | Rapid movement: | direct error message |

I2= Defines which value is still allowed for the maximum unbalance. If this is not programmed the value in MC454 'Unbalance monitor 1: limit' is taken. The value lies between 0 and 100 [µm].

I3= Defines the maximum sum (of unbalances exceeding the limit) before an alarm is issued. If this is not programmed the value in MC454 'Unbalance monitor 1: sum over limit' is taken. The value lies between 0 and 1000 [µm].

Note:

- G228 is only present when MC314 'milling and turning mode' is active.
- G228 activates the first unbalance monitor. The setting of the 1st unbalance monitor is taken from the machine constants MC454 and MC455 or, if programmed, from parameters I2= and I3=. Depending on parameter I1=, an error message is issued.

SWITCHING OFF THE UNBALANCE MONITOR (G227)

Note:

- G227 switches off G228 and therefore the 1st unbalance monitor.
- G227 is automatically activated after <Reset control>, <Cancel program> or M30.
- The 2nd unbalance monitor cannot be switched off.

OPERATOR INTERFACE

The current unbalance value is displayed in the Spindle performance display. Here the 1st programmable limit is marked in yellow and the second fixed limit is marked in red. The highest unbalance value that has occurred since the start of the program or programming of G228 is shown in green.

The display is only present when one of the unbalance monitors is active. The red marking is always 90% along the total length.

ERROR MESSAGES

S228 Unbalance monitor 1: Excess unbalance

Class: D

The 1st unbalance monitor generates an alarm. Whether and when this error occurs depends on the machine constants MC454 and MC455 and/or can be programmed in G228 'Unbalance monitor: ON'

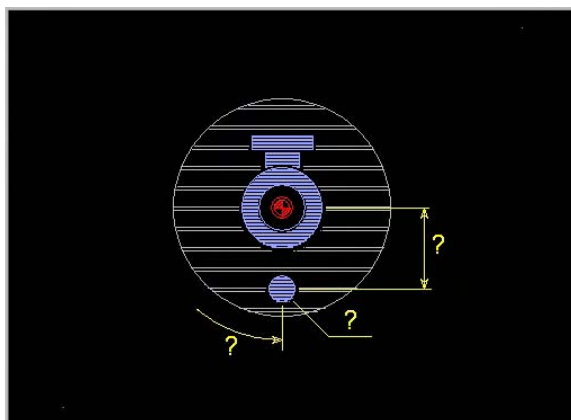
S229 Unbalance monitor 2: Excess unbalance

Class: D

The 2nd unbalance monitor generates an alarm. Whether and when this error occurs depends on the machine constants MC456 and MC457.

19.12.4 G691 Measure unbalance

This cycle calculates the instantaneous unbalance. It gives the operator a suggestion how to compensate for the unbalance. This cycle should be called after each clamping operation and after milling mode..



```
G  Unbalance measurement
D  Speed limitation      [rev/min]
```

- D Maximum speed for terminating the measurement
 Basic setting MC2691 'maximum speed
 Minimum value 50 [rpm]
 The speed limit should be at least as high as the programmed speed for turning machining.

Notes and application

When detecting unbalance, the position error of the linear axis is measured with rising speed. The speed is increased in steps of 25 rpm. When the position error has reached the maximum value (MC451) or the maximum speed has been reached, the measurement is terminated. The unbalance is calculated from the measured error and the stored calibration data.

The unbalance (gmm) and compensation position (degrees) are displayed. This position is approached at the end of the cycle.

Example: Balancing a workpiece

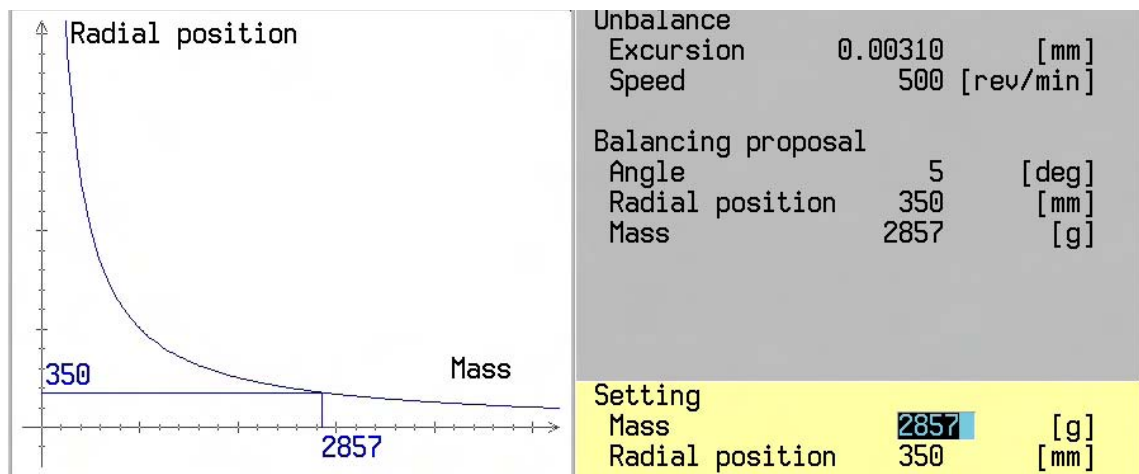
Nxx G691 D500

Explanation:

1. Start balancing cycle with maximum speed of 500 rpm.
2. Unbalance is measured. Calculated mass and radial position (distance and angle) are shown in the window. The balance position is automatically positioned.
3. Enter the weight of an available mass in the window.
4. The CNC displays the new radial distance for the available mass.
5. Fit the mass at the radial position (distance and angle). Terminate with start.
6. Check the balance quality by repeating the balancing cycle G691. The unbalance mass must be very small. If necessary, balance again with the displayed mass.

Representation of measurement results

Once the unbalance detection measurement is terminated, the measurement results are displayed instead of the input and support fields. This image is created by G350.



Left:

The relationship between mass and position are shown graphically.

Top right:

The measured unbalance causes a deflection at the speed displayed. This unbalance can be compensated in accordance with the balancing suggestion.

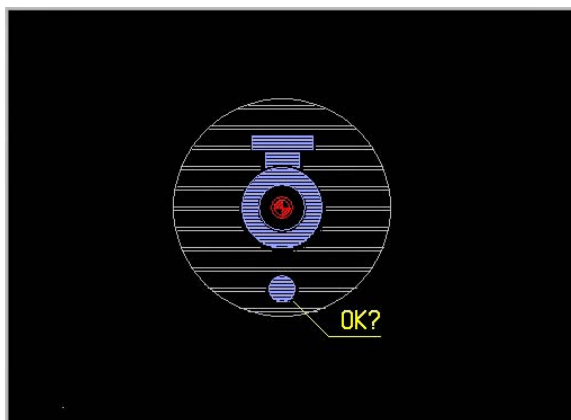
Bottom right:

The radial position for a selected mass is calculated in the dialog window. The calculation takes place after pressing the <ENTER> key. The START key terminates the cycle and closes this window.

In **automatic mode**, the left graphical window is not shown so that the program pointer remains visible.

19.12.5 G692 Unbalance checking

This cycle checks that the unbalance does not exceed a particular value. It should be called at the start of every turning operation to ensure that the concentric error does not exceed the desired tolerance or the specified limit.



```
G   Unbalance check
C1= Allowed excursion    [mm|inch]
D   Check speed         [rev/min]
```

- C1= Maximum unbalance for message
Basic setting MC451 "maximum deflection".
- D Programmed speed for checking
Basic setting MC2691 "maximum speed"

Notes and application

When checking unbalance, the deflection of the linear axis is measured at a specified speed. If the deflection reaches the value C1=, an error message is issued.

Example: Checking unbalance.

G692 C1=0.003 D500 The CNC detects whether the deflection of the table is within the limit of 0.003 mm at a speed of 500 rpm. If the deflection is greater than the value entered (C1=), the program is stopped.

Unbalance example

Program example	Description
N9999	
N1 G691 D500	1 Start balancing cycle with maximum speed of 500 rpm. 2 Unbalance is measured. Calculated mass and radial position (distance and angle) are shown in the window. The balance position is automatically located. 3 Enter the weight of an available mass in the window. 4 The CNC displays the new radial distance for the available mass. 5 Fit the mass at the radial position (distance and angle). Terminate with start.
N2 G691 D500	Check the balance quality by repeating the balancing cycle G691. The unbalance mass must be very small. If necessary, balance again with the displayed mass.
N...	Milling Unbalance may change due to milling processes or changes in the clamping.
N30 G37	Start turning mode
N31 G692 D500	Check whether unbalance is still correct
N...	Turning

19.13 Turning cycles

AVAILABILITY

The machine builder must prepare machine and CNC for turning operations. If your machine is not equipped with all the G functions described here, please refer to your machine manual.

The turning cycles are executed as macros, every block can be seen in the display and each block is processed as a single block.

General notes and application

STARTING POINT

The starting point determines the place where the tool starts machining. The cutting steps start from this position. If the tool is a long distance away, several cutting steps take place. If the tool is between Y1= and Y2=, cutting will start there and the cutting may not all be carried out.

If the co-ordinate of the starting point Y is smaller than the co-ordinate of the machining starting point Y1, the machine first travels to co-ordinate Z1.

TOOL MEMORY ADDRESSES

The following addresses are used in the tool memory:

C Tool tip radius

O Tool orientation

C6 Tool width (Grooving cycles)

If no O is entered in the tool memory, a standard orientation is assumed depending on the machining.

RADIUS COMPENSATION

Tool tip radius compensation is carried out automatically in this G function.

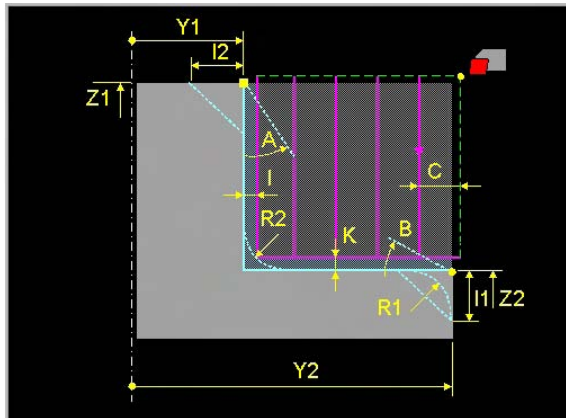
Cycle survey

Clearance, Grooving, Undercut and threading cycles

The control system offers several clearance- and grooving cycles. The clearance cycles are divided into two groups: clearance- and roughing cycles

Cycles	Cycle	G-Function
Clearance	Clearance axial	G822
	Clearance radial	G823
	Clearance axial finishing	G826
	Clearance radial finishing	G827
Roughing	Roughing axial	G832
	Roughing radial	G833
	Roughing axial finishing	G836
	Roughing radial finishing	G837
Grooving (Standard)	Grooving axial	G842
	Grooving radial	G843
	Grooving axial finishing	G847
	Grooving radial finishing	G846
Grooving (Universal)	Grooving axial -Universal	G844
	Grooving radial -Universal	G845
	Grooving axial Finishing -Universal	G848
	Grooving radial Finishing -Universal	G849
Undercut	Undercut DIN 76	G850
	Undercut DIN 509 E	G851
	Undercut DIN 509 F	G852
Threading	Threading Axial	G861
	Threading Conical	G862

19.13.1 G822 Clearance axial



G Clearance axial
 Y Starting point
 Z Starting point
 Y1= Beginpoint contour
 Z1= Beginpoint contour
 Y2= Endpoint contour
 Z2= Endpoint contour
 C Cutting depth
 A Angle 1
 B Angle 2
 I1= Chamfer length 1
 R1= Radius 1
 I2= Chamfer length 2
 R2= Radius 2
 I Finishing

K Finishing
 S1= (Cutting) Speed
 F Feed

Y	Starting point.	Position of tool in radial direction. This position is the starting point for machining. Y is reduced with C until Y1= is reached.
Z	Starting point.	Position of tool in axial direction. This position is the starting point for machining. Machining starts at Z until Z2 is reached.
Y1=	Contour starting point	Starting point of the contour to be machined.
Z1=	Contour starting point	Starting point of the contour to be machined.
Y2=	Contour end point	End point of the contour to be machined.
Z2=	Contour end point	End point of the contour to be machined.
C	Radial feed depth	Dimension by which the tool is fed in the radial direction in each case. The depth does not have to be a multiple of the feed depth.
A	Angle	Basic setting A=0. Angle (>0) at contour starting point. Angle A or B must be chosen so that the tool does not undercut.
B	Angle	Basic setting B=0. Angle (>0) at contour end point.
I1=	Chamfer length	Basic setting I1=0. Chamfer length at contour end point. Only I1= or R1= may be programmed.
R1=	Rounding	Basic setting R1=0. Rounding at contour end point.
I2=	Chamfer length	Basic setting I2=0. Chamfer length at contour starting point.
R2=	Rounding	Basic setting R2= tool tip radius.
I and K		Rounding between angles A and B. Stock removal

Basic settings

A=0, B=0, I1=0, R1=0, I2=0, R2= Tool nose radius, I=0, K=0

Associated functions

G827 for finish machining

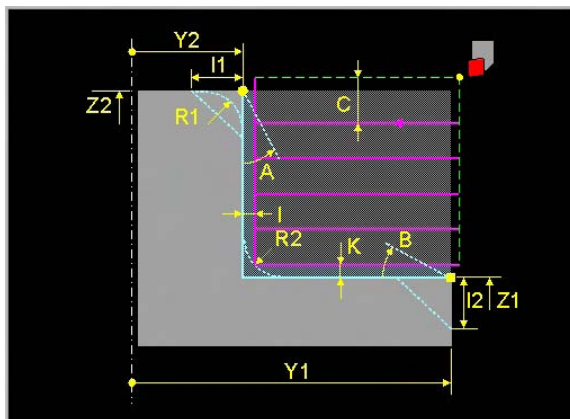
Notes and application

Cutting takes place first, then finish machining.

Tool orientation may only be 4, 5 or 6.

The tool path is corrected for the tip radius.

19.13.2 G823 Clearance radial



G Clearance radial
 Y Starting point
 Z Starting point
 Y1= Beginpoint contour
 Z1= Beginpoint contour
 Y2= Endpoint contour
 Z2= Endpoint contour
 C Cutting depth
 A Angle 1
 B Angle 2
 I1= Chamfer length 1
 R1= Radius 1
 I2= Chamfer length 2
 R2= Radius 2
 I Finishing

K Finishing
 S1= (Cutting) Speed
 F Feed

Y	Starting point.	Position of tool in radial direction. This position is the starting point for machining. Machining starts at Y until Y2 is reached.
Z	Starting point.	Position of tool in axial direction. This position is the starting point for machining. Z is reduced with C until Z1= is reached.
Y1=	Contour starting point	Starting point of the contour to be machined.
Z1=	Contour starting point	Starting point of the contour to be machined.
Y2=	Contour end point	End point of the contour to be machined.
Z2=	Contour end point	End point of the contour to be machined.
C	Radial feed depth	Dimension (incremental: by which the tool is fed in the axial direction in each case. The depth does not have to be a multiple of the feed depth.
A	Angle	Basic setting A=0. Angle (>0) at contour starting point. Angle A or B must be chosen so that the tool does not undercut.
B	Angle	Basic setting B=0. Angle (>0) at contour end point.
I1=	Chamfer length	Basic setting I1=0. Chamfer length at contour end point. Only I1= or R1= may be programmed.
R1=	Rounding	Basic setting R1=0. Rounding at contour end point.
I2=	Chamfer length	Basic setting I2=0. Chamfer length at contour starting point.
R2=	Rounding	Basic setting R2= tool tip radius. Rounding between angles A and B.
I and K		Stock removal

Basic settings

A=0, B=0, I1=0, R1=0, I2=0, R2= Tool nose radius, I=0, K=0

Associated functions

G827 for finish machining

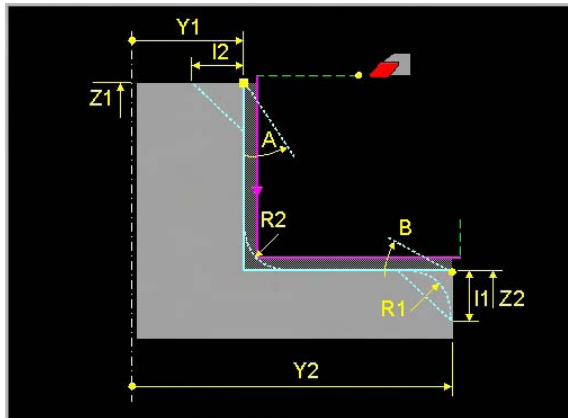
Notes and application

Cutting takes place first, then finish machining.

Tool orientation may only be 4, 5 or 6.

The tool path is corrected for the tip radius.

19.13.3 G826 Clearance axial finishing



```

G   Clearance axial finishing
Y   Starting point
Z   Starting point
Y1= Beginpoint contour
Z1= Beginpoint contour
Y2= Endpoint contour
Z2= Endpoint contour
A   Angle 1
B   Angle 2
I1= Chamfer length 1
R1= Radius 1
I2= Chamfer length 2
R2= Radius 2
S1= (Cutting) Speed
F   Feed

```

Y	Starting point.	Position of tool in radial direction. This position is the starting point for finish machining.
Z	Starting point.	Position of tool in axial direction. This position is the starting point for finish machining. Finish machining starts at Y.
Y1=	Contour starting point	Starting point of the contour to be machined.
Z1=	Contour starting point	Starting point of the contour to be machined.
Y2=	Contour end point	End point of the contour to be machined.
Z2=	Contour end point	End point of the contour to be machined.
A	Angle	Basic setting A=0. Angle (>0) at contour starting point. Angle A or B must be chosen so that the tool does not undercut.
B	Angle	Basic setting B=0. Angle (>0) at contour end point.
I1=	Chamfer length	Basic setting I1=0. Chamfer length at contour end point. Only I1= or R1= may be programmed.
R1=	Rounding	Basic setting R1=0. Rounding at contour end point.
I2=	Chamfer length	Basic setting I2=0. Chamfer length at contour starting point.
R2=	Rounding	Basic setting R2= tool tip radius. Rounding between angles A and B.

Basic settings

A=0, B=0, I1=0, I2=0, R2= Tool nose radius

Associated functions

G822 for rough machining

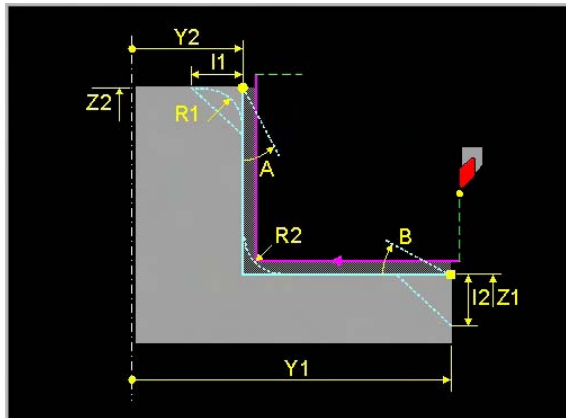
Notes and application

Finish machining goes from Y1/Z1 to Y2/Z2.

Tool orientation may only be 4, 5 or 6.

The tool path is corrected for the tip radius.

19.13.4 G827 Clearance radial finishing



```

G   Clearance radial finishing
Y   Starting point
Z   Starting point
Y1= Beginpoint contour
Z1= Beginpoint contour
Y2= Endpoint contour
Z2= Endpoint contour
A   Angle 1
B   Angle 2
I1= Chamfer length 1
R1= Radius 1
I2= Chamfer length 2
R2= Radius 2
S1= (Cutting) Speed
F   Feed

```

Y	Starting point.	Position of tool in radial direction. This position is the starting point for finish machining. Finish machining starts at Y until Y2 is reached.
Z	Starting point.	Position of tool in axial direction. This position is the starting point for finish machining.
Y1=	Contour starting point	Starting point of the contour to be machined.
Z1=	Contour starting point	Starting point of the contour to be machined.
Y2=	Contour end point	End point of the contour to be machined.
Z2=	Contour end point	End point of the contour to be machined.
A	Angle	Basic setting A=0. Angle (>0) at contour starting point. Angle A or B must be chosen so that the tool does not undercut.
B	Angle	Basic setting B=0. Angle (>0) at contour end point.
I1=	Chamfer length	Basic setting I1=0. Chamfer length at contour end point. Only I1= or R1= may be programmed.
R1=	Rounding	Basic setting R1=0. Rounding at contour end point.
I2=	Chamfer length	Basic setting I2=0. Chamfer length at contour starting point.
R2=	Rounding	Basic setting R2= tool tip radius. Rounding between angles A and B.

Basic settings

A=0, B=0, I1=0, R1=0, I2=0, R2= Tool nose radius

Associated functions

G823 for rough machining

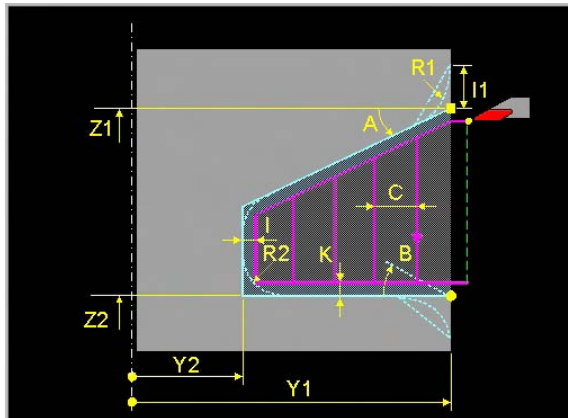
Notes and application

Finish machining goes from Y1/Z1 to Y2/Z2.

Tool orientation may only be 4, 5 or 6.

The tool path is corrected for the tip radius

19.13.5 G832 Roughing axial



G Roughing axial
 Y Starting point
 Z Starting point
 Y1= Beginpoint contour
 Z1= Beginpoint contour
 Y2= Endpoint contour
 Z2= Endpoint contour
 C Cutting depth
 A Angle 1
 B Angle 2
 I1= Chamfer length 1
 R1= Radius 1
 R2= Radius 2
 I Finishing
 K Finishing

S1= (Cutting) Speed
 F Feed

Y	Starting point.	Position of tool in Radial direction. This position is the starting point for machining. Machining starts at Y and is reduced with C until Y2= is reached.
Z	Starting point.	Position of tool in axial direction. This position is the starting point for machining. Machining starts at Z1= until Z2= is reached.
Y1=	Contour starting point	Starting point of the contour to be machined.
Z1=	Contour starting point	Starting point of the contour to be machined.
Y2=	Contour end point	End point of the contour to be machined.
Z2=	Contour end point	End point of the contour to be machined.
C	Radial feed depth	Dimension by which the tool is fed in the radial direction in each case. The depth does not have to be a multiple of the feed depth.
A	Angle	Basic setting A=0. Angle (>0) at contour starting point. (Z1=)
B	Angle	Angles A and B must be chosen so that the tool does not undercut. Basic setting B=0. Angle (>0) at contour end point. (Z2=)
I1=	Chamfer length	Basic setting I1=0. Chamfer length at start and end of contour. Only I1= or R1= may be programmed.
R1=	Rounding	Basic setting R1=0. Rounding at start and end of contour.
R2=	Rounding	Basic setting R2= tool tip radius. Rounding at the bottom of the contour.
I and K		Stock removal

Basic settings

A=0, B=0, I1=0, R1=0, R2= Tool nose radius, I=0, K=0

Associated functions

G837 for finish machining

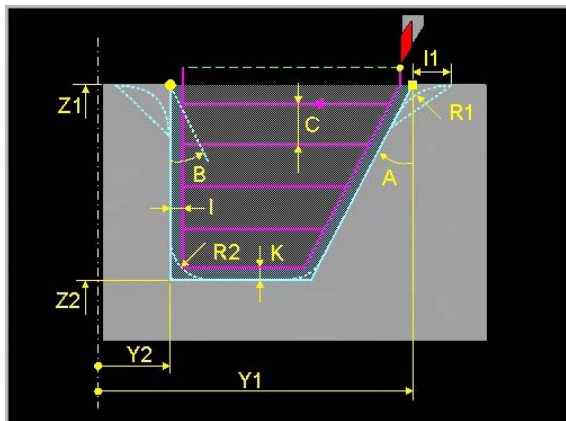
Notes and application

Rough cutting takes place first, then finish machining.

Tool orientation may only be 3, 4 or 5.

The tool path is corrected for the tip radius.

19.13.6 G833 Roughing radial



G Roughing radial
 Y Starting point
 Z Starting point
 Y1= Beginpoint contour
 Z1= Beginpoint contour
 Y2= Endpoint contour
 Z2= Endpoint contour
 C Cutting depth
 A Angle 1
 B Angle 2
 I1= Chamfer length 1
 R1= Radius 1
 R2= Radius 2
 I Finishing
 K Finishing

S1= (Cutting) Speed
 F Feed

Y	Starting point.	Position of tool in radial direction. This position is the starting point for machining. Machining starts at Y1= until Y2= is reached.
Z	Starting point.	Position of tool in radial direction. This position is the starting point for machining. Machining starts at Z and is reduced with C until Z2= is reached.
Y1=	Contour starting point	Starting point of the contour to be machined.
Z1=	Contour starting point	Starting point of the contour to be machined.
Y2=	Contour end point	End point of the contour to be machined.
Z2=	Contour end point	End point of the contour to be machined.
C	Radial feed depth	Dimension (incremental) by which the tool is fed in the axial direction in each case. The depth does not have to be a multiple of the feed depth.
A	Angle	Basic setting A=0. Angle (>0) at contour starting point. (Y1=)
B	Angle:	Angles A and B must be chosen so that the tool does not undercut. Basic setting B=0. Angle (>0) at contour end point. (Y2=)
I1=	Chamfer length	Basic setting I1=0. Chamfer length at start and end of contour. Only I1= or R1= may be programmed.
R1=	Rounding	Basic setting R1=0. Rounding at start and end of contour.
R2=	Rounding	Basic setting R2= tool tip radius. Rounding at the bottom of the contour.
I and K		Stock removal

Basic settings

A=0, B=0, I1=0, R1=0, R2= Tool nose radius, I=0 K=0

Associated functions

G837 for finish machining

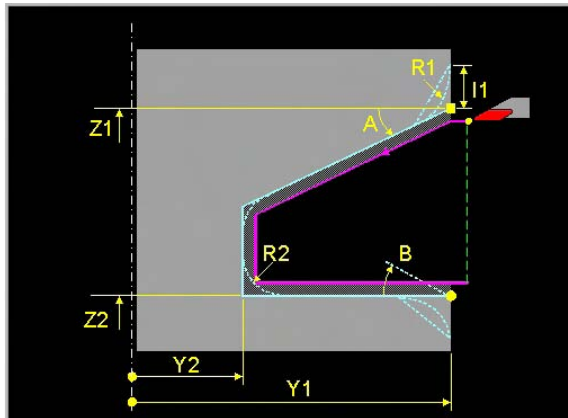
Notes and application

Rough cutting takes place first, then finish machining.

Tool orientation may only be 5, 6 or 7.

The tool path is corrected for the tip radius.

19.13.7 G836 Roughing axial finishing



```

G   Roughing axial finishing
Y   Starting point
Z   Starting point
Y1= Beginpoint contour
Z1= Beginpoint contour
Y2= Endpoint contour
Z2= Endpoint contour
A   Angle 1
B   Angle 2
I1= Chamfer length 1
R1= Radius 1
R2= Radius 2
S1= (Cutting) Speed
F   Feed

```

Y	Starting point.	Position of tool in radial direction. This position is the starting point for finish machining.
Z	Starting point.	Position of tool in axial direction. This position is the starting point for finish machining. Finish machining starts at Z1= until Z2= is reached.
Y1=	Contour starting point	Starting point of the contour to be machined.
Z1=	Contour starting point	Starting point of the contour to be machined.
Y2=	Contour end point	End point of the contour to be machined.
Z2=	Contour end point	End point of the contour to be machined.
A	Angle	Basic setting A=0. Angle (>0) at contour starting point. (Z1=)
B	Angle	Angles A and B must be chosen so that the tool does not undercut. Basic setting B=0. Angle (>0) at contour end point. (Z2=)
I1=	Chamfer length	Basic setting I1=0. Chamfer length at start and end of contour. Only I1= or R1= may be programmed.
R1=	Rounding	Basic setting R1=0. Rounding at start and end of contour.
R2=	Rounding	Basic setting R2= tool tip radius. Rounding at the bottom of the contour.

Basic settings

A=0, B=0, I1=0, R1=0, R2= Tool nose radius

Associated functions

G832 for finish machining

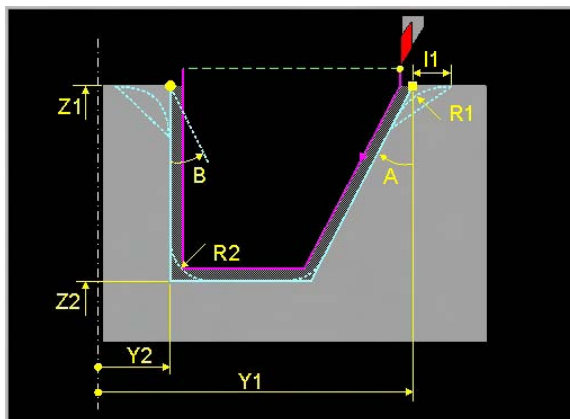
Notes and application

Finish machining goes from Y1/Z1 to Y1/Z2.

Tool orientation may only be 3, 4 or 5.

The tool path is corrected for the tip radius.

19.13.8 G837 Roughing radial finishing



```

G   Roughing radial finishing
Y   Starting point
Z   Starting point
Y1= Beginpoint contour
Z1= Beginpoint contour
Y2= Endpoint contour
Z2= Endpoint contour
A   Angle 1
B   Angle 2
I1= Chamfer length 1
R1= Radius 1
R2= Radius 2
S1= (Cutting) Speed
F   Feed

```

Y	Starting point.	Position of tool in radial direction. This position is the starting point for finish machining. Finish machining starts at Y1= until Y2= is reached.
Z	Starting point.	Position of tool in radial direction. This position is the starting point for finish machining.
Y1=	Contour starting point	Starting point of the contour to be machined.
Z1=	Contour starting point	Starting point of the contour to be machined.
Y2=	Contour end point	End point of the contour to be machined.
Z2=	Contour end point	End point of the contour to be machined.
A	Angle	Basic setting A=0. Angle (>0) at contour starting point. (Y1=)
B	Angle	Angles A and B must be chosen so that the tool does not undercut.
I1=	Chamfer length	Basic setting B=0. Angle (>0) at contour end point. (Y2=)
		Basic setting I1=0. Chamfer length at start and end of contour.
		Only I1= or R1= may be programmed.
R1=	Rounding	Basic setting R1=0. Rounding at start and end of contour.
R2=	Rounding	Basic setting R2= tool tip radius. Rounding at the bottom of the contour.

Basic settings

A=0, B=0, I1=0, R1=0, R2= Tool nose radius

Associated functions

G833 for finish machining

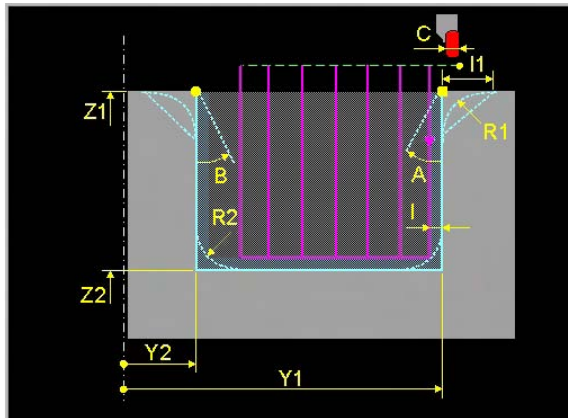
Notes and application

Finish machining goes from Y1/Z1 to Y2/Z1.

Tool orientation may only be 5, 6 or 7.

The tool path is corrected for the tip radius..

19.13.9 G842 Grooving axial



G Grooving axial
 Y Starting point
 Z Starting point
 Y1= Beginpoint contour
 Z1= Beginpoint contour
 Y2= Endpoint contour
 Z2= Endpoint contour
 C Tool width
 A Angle 1
 B Angle 2
 I1= Chamfer length 1
 R1= Radius 1
 R2= Radius 2
 I Finishing
 S1= (Cutting) Speed

F Feed

Y	Starting point.	Position of tool in radial direction. This position is the starting point for machining.
Z	Starting point.	Position of tool in axial direction. This position is the starting point for machining.
Y1=	Contour starting point	Starting point of the contour to be machined.
Z1=	Contour starting point	Starting point of the contour to be machined.
Y2=	Contour end point	End point of the contour to be machined.
Z2=	Contour end point	End point of the contour to be machined.
C	Chisel width	Width of tool. The feed width is C minus twice the tip radius
A	Angle	Basic setting A=0. Angle (>0) at contour starting point. (Y1=)
B	Angle	Basic setting B=0. Angle (>0) at contour end point. (Y2=)
I1=	Chamfer length	Basic setting I1=0. Chamfer length at start and end of contour. Only I1= or R1= may be programmed.
R1=	Rounding	Basic setting R1=0. Rounding at start and end of contour.
R2=	Rounding	Basic setting R2= tool corner radius. Rounding at the bottom of the contour. Finish machining allowance: basic setting I=0.
I		Stock removal

Basic settings

A=0, B=0, I1=0, R1=0, R2= Tool nose radius, I=0

Associated functions

G846 for finish machining

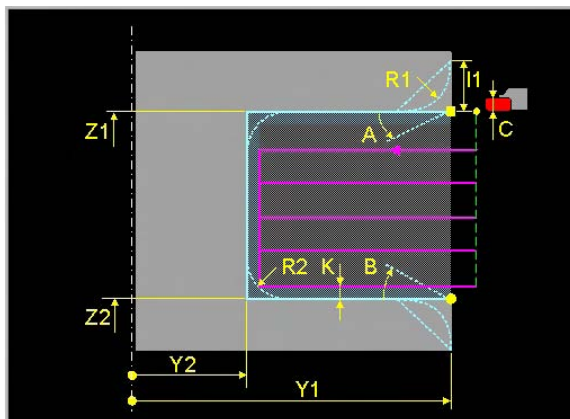
Notes and application

Rough cutting takes place first, then finish machining.

Tool orientation may only be 5, 6 or 7.

The tool path is corrected for the tip radius..

19.13.10 G843 Grooving radial



G Grooving radial
 Y Starting point
 Z Starting point
 Y1= Beginpoint contour
 Z1= Beginpoint contour
 Y2= Endpoint contour
 Z2= Endpoint contour
 C Tool width
 A Angle 1
 B Angle 2
 I1= Chamfer length 1
 R1= Radius 1
 R2= Radius 2
 K Finishing
 S1= (Cutting) Speed

F Feed

Y	Starting point.	Position of tool in radial direction. This position is the starting point for machining. Machining starts at Y until Y2 is reached.
Z	Starting point.	Position of tool in axial direction. This position is the starting point for machining. Machining starts at Z2= with the feed width until Z1= is reached.
Y1=	Contour starting point	Starting point of the contour to be machined.
Z1=	Contour starting point	Starting point of the contour to be machined.
Y2=	Contour end point	End point of the contour to be machined.
Z2=	Contour end point	End point of the contour to be machined.
C	Chisel width	Width of tool. The feed width is C minus twice the tip radius
A	Angle	Basic setting A=0. Angle (>0) at contour starting point. (Z1=)
B	Angle	Basic setting B=0. Angle (>0) at contour end point. (Z2=)
I1=	Chamfer length	Basic setting I1=0. Chamfer length at start and end of contour. Only I1= or R1= may be programmed.
R1=	Rounding	Basic setting R1=0. Rounding at start and end of contour.
R2=	Rounding	Basic setting R2= tool tip radius. Rounding at the bottom of the contour.
K		Stock removal

Basic settings

A=0, B=0, I1=0, R1=0, R2= Tool nose radius, K=0

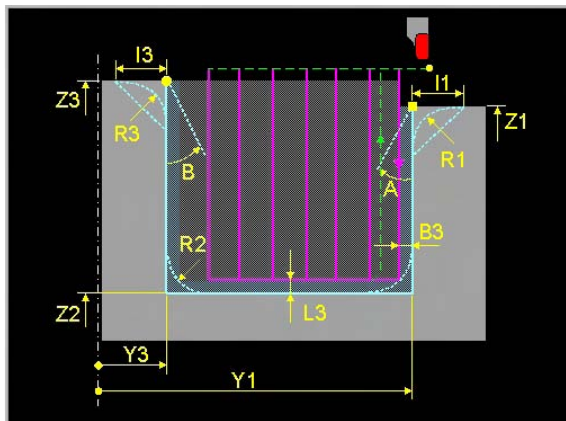
Associated functions

G847 for finish machining

Notes and application

Rough cutting takes place first, then finish machining.
 Tool orientation may only be 3, 4 or 5.
 The tool path is corrected for the tip radius.

19.13.11 G844 Grooving universal axial roughing



G Grooving axial universal
 Y Starting point
 Z Starting point
 Y1= Beginpoint contour
 Z1= Beginpoint contour
 R2= Radius 2
 Z2= Groove depth
 Y3= Endpoint contour
 Z3= Endpoint contour
 A Angle 1
 B Angle 2
 I1= Chamfer length 1
 I3= Chamfer length 3
 R1= Radius 1
 R3= Radius 3

B3= Finishing allowance
 L3= Finishing allowance
 I7= Finishing 0=no 1=yes
 S1= (Cutting) Speed
 F Feed

Y, Z Starting point grooving cycle.
 Y1=, Z1= Contour Starting point
 Z2= Contour bottom
 Y3=, Z3= Contour end point. If Z3 is not programmed then (Z3=Z1)
 A Angle (0-89°) at groove starting point (Y1, Z1)
 B Angle (0-89°) at groove end point (Y3, Z3)
 I1= Chamfer length at groove starting point (Y1, Z1)
 I3= Chamfer at groove end point (Y3, Z3)
 R1= Rounding at groove starting point. (Y1, Z1)
 R2= Rounding at both sides of groove bottom.
 R3= Rounding at groove end point (Y3, Z3)
 B3= Finishing allowance along the Z-Axis
 L3= Finishing allowance along the Y-Axis
 I7= Finishing included 0=No 1=Yes

Basic settings: A=0, B= 0, I1=0, R1= 0, I3=0, R3=0, R2=0, I7=0, B3=0, L3= 0

Associated functions: G848 for finishing

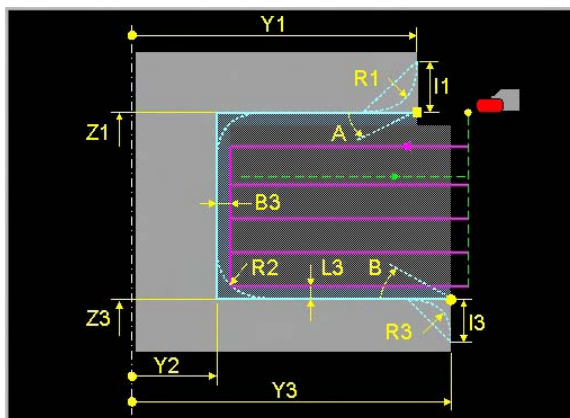
Notes and application

- First grooving (roughing) than, depending on (I7), finishing.
- The tool width (C6) is taken from the tool table. An error code appears if the tool width is not available.
- Groove displacement is (C6-2xC). Maximum displacement is (C6)
- Tool orientation (O):
 - . The tool orientation is stored in the tool table
 - . With the G-function G302, the tool orientation can be overwritten in the program
 - . If there is no tool orientation available, the tool orientation will be calculated from the cycle (sense of machining).
- At the end of the groove, the tool is retracted at an angle of 45° and 0.5 mm away from the groove side

Remark:

Make sure that the tool orientation physically corresponds with the actual tool position: Left/Right or In/Outside cutting edge.

19.13.12 G845 Grooving universal radial roughing



G Grooving radial universal
 Y Starting point
 Z Starting point
 Y1= Beginpoint contour
 Z1= Beginpoint contour
 Y2= Groove depth
 R2= Radius 2
 Y3= Endpoint contour
 Z3= Endpoint contour
 A Angle 1
 B Angle 2
 I1= Chamfer length 1
 I3= Chamfer length 3
 R1= Radius 1
 R3= Radius 3

B3= Finishing allowance
 L3= Finishing allowance
 I7= Finishing 0=no 1=yes
 S1= (Cutting) Speed
 F Feed

Y, Z Starting point grooving cycle.
 Y1=, Z1= Contour Starting point
 Y2= Contour bottom
 Y3=, Z3= Contour end point. If Y3 is not programmed then (Y3=Y1)
 A Angle (0-89°) at groove starting point (Y1, Z1)
 B Angle (0-89°) at groove end point (Y3, Z3)
 I1= Chamfer length at groove starting point (Y1, Z1)
 I3= Chamfer at groove end point (Y3, Z3)
 R1= Rounding at groove starting point. (Y1, Z1)
 R2= Rounding at both sides of groove bottom.
 R3= Rounding at groove end point (Y3, Z3)
 B3= Finishing allowance along the Y-Axis
 L3= Finishing allowance along the Z-Axis
 I7= Finishing included 0=No 1=Yes

Basic settings: A=0, B= 0, I1=0, R1= 0, I3=0, R3=0, R2=0, I7=0, B3=0, L3= 0

Associated functions: G848 for finishing

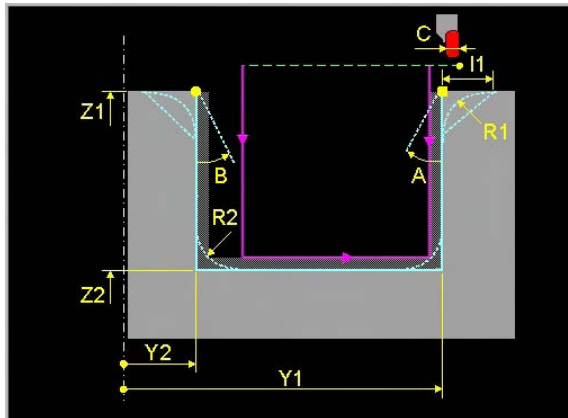
Notes and application

- First grooving (roughing) than, depending on (I7), finishing.
- The tool width (C6) is taken from the tool table. An error code appears if the tool width is not available.
- Groove displacement is (C6-2xC). Maximum displacement is (C6)
- Tool orientation (O):
 - . The tool orientation is stored in the tool table
 - . With the G-function G302, the tool orientation can be overwritten in the program
 - . If there is no tool orientation available, the tool orientation will be calculated from the cycle (sense of machining).
- At the end of the groove, the tool is retracted at an angle of 45° and 0.5 mm away from the groove side

Remark:

Make sure that the tool orientation physically corresponds with the actual tool position: Left/Right or In/Outside cutting edge.

19.13.13 G846 Grooving axial finishing



G Grooving axial finish
 Y Starting point
 Z Starting point
 Y1= Beginpoint contour
 Z1= Beginpoint contour
 Y2= Endpoint contour
 Z2= Endpoint contour
 C Tool width
 A Angle 1
 B Angle 2
 I1= Chamfer length 1
 R1= Radius 1
 R2= Radius 2
 S1= (Cutting) Speed
 F Feed

F Feed

Y	Starting point.	Position of tool in radial direction. This position is the starting point for machining. Machining starts at Y until Y2 is reached.
Z	Starting point.	Position of tool in axial direction. This position is the starting point for at Z2= until Z1= is reached.
Y1=	Contour starting point	Starting point of the contour to be machined.
Z1=	Contour starting point	Starting point of the contour to be machined.
Y2=	Contour end point	End point of the contour to be machined.
Z2=	Contour end point	End point of the contour to be machined.
C	Chisel width	Width of tool. The feed width is C minus twice the corner radius
A	Angle	Basic setting A=0. Angle (>0) at contour starting point. (Y1=)
B	Angle	Basic setting B=0. Angle (>0) at contour end point. (Y2=)
I1=	Chamfer length	Basic setting I1=0. Chamfer length at start and end of contour. Only I1= or R1= may be programmed.
R1=	Rounding	Basic setting R1=0. Rounding at start and end of contour.
R2=	Rounding	Basic setting R2= tool tip radius. Rounding at the bottom of the contour.
I		Stock removal

Basic settings

A=0, B=0, I1=0, R1=0, R2= Tool nose radius, I=0

Associated functions

G842 for finish machining

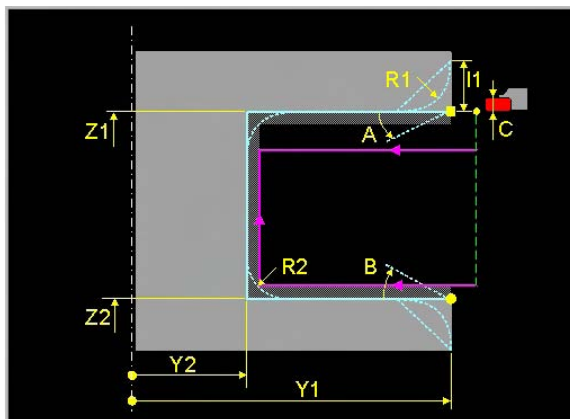
Notes and application

Finish machining goes from Y1/Z1 to Y1/Z2.

Tool orientation may only be 5, 6 or 7.

The tool path is corrected for the tip radius.

19.13.14 G847 Grooving radial finishing



G Grooving radial finish
 Y Starting point
 Z Starting point
 Y1= Beginpoint contour
 Z1= Beginpoint contour
 Y2= Endpoint contour
 Z2= Endpoint contour
 C Tool width
 A Angle 1
 B Angle 2
 I1= Chamfer length 1
 R1= Radius 1
 R2= Radius 2
 S1= (Cutting) Speed
 F Feed

F Feed

Y	Starting point.	Position of tool in radial direction. This position is the starting point for finish machining. Finish machining starts at Y until Y2 is reached.
Z	Starting point.	Position of tool in axial direction. This position is the starting point for finish machining.
Y1=	Contour starting point	Starting point of the contour to be machined.
Z1=	Contour starting point	Starting point of the contour to be machined.
Y2=	Contour end point	End point of the contour to be machined.
Z2=	Contour end point	End point of the contour to be machined.
C	Chisel width	Width of tool. The feed width is C minus twice the corner radius
A	Angle	Basic setting A=0. Angle (>0) at contour starting point. (Z1=)
B	Angle	Basic setting B=0. Angle (>0) at contour end point. (Z2=)
I1=	Chamfer length	Basic setting I1=0. Chamfer length at start and end of contour. Only I1= or R1= may be programmed.
R1=	Rounding	Basic setting R1=0. Rounding at start and end of contour.
R2=	Rounding	Basic setting R2= tool tip radius. Rounding at the bottom of the contour.
K		Stock removal

Basic settings

A=0, B=0, I1=0, R1=0, R2= Tool nose radius, K=

Associated functions

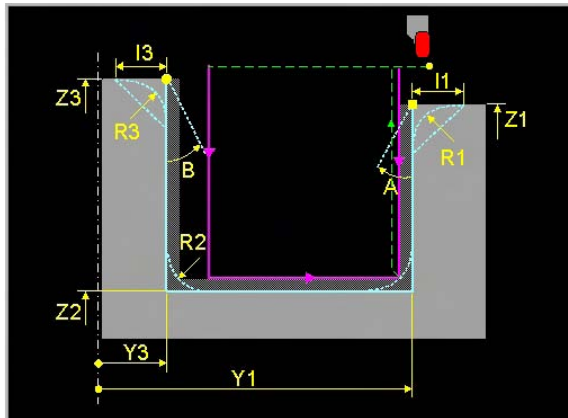
G843 for rough machining

Notes and application

Finish machining goes from Y1/Z2 to Y1/Z1.

Tool orientation may only be 3, 4 or 5.

The tool path is corrected for the tip radius

19.13.15 G848 Grooving universal axial, finishing

```

G   Grooving axial universal finish
Y   Starting point
Z   Starting point
Y1= Beginpoint contour
Z1= Beginpoint contour
R2= Radius 2
Z2= Groove depth
Y3= Endpoint contour
Z3= Endpoint contour
A   Angle 1
B   Angle 2
I1= Chamfer length 1
I3= Chamfer length 3
R1= Radius 1
R3= Radius 3

```

```

S1= (Cutting) Speed
F   Feed

```

Y, Z Starting point grooving cycle.
 Y1=, Z1= Contour Starting point
 Z2= Contour bottom
 Y3=, Z3= Contour end point. If Z3 is not programmed then (Z3=Z1)
 A Angle (0-89°) at groove starting point (Y1, Z1)
 B Angle (0-89°) at groove end point (Y3, Z3)
 I1= Chamfer length at groove starting point (Y1, Z1)
 I3= Chamfer at groove end point (Y3, Z3)
 R1= Rounding at groove starting point. (Y1, Z1)
 R2= Rounding at both sides of groove bottom.
 R3= Rounding at groove end point (Y3, Z3)

Basic settings: A=0, B= 0, I1=0, R1= 0, I3=0, R3=0, R2=0

Associated functions: G844 for roughing

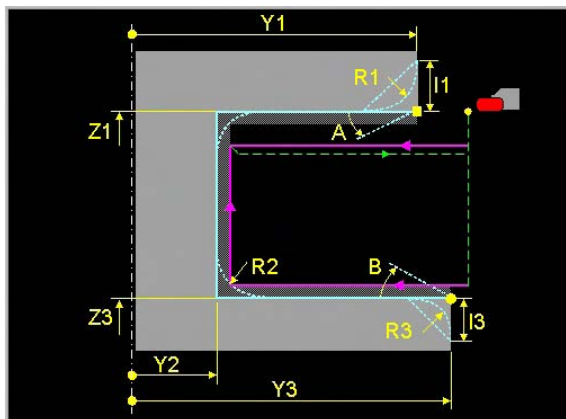
Notes and application

- First the opposite groove side is cut, than the adjoining groove side followed by the groove bottom
- The tool width (C6) is taken from the tool table. An error code appears if the tool width is not available.
- Tool orientation (O):
 - . The tool orientation is stored in the tool table
 - . With the G-function G302, the tool orientation can be overwritten in the program
 - . If there is no tool orientation available, the tool orientation will be calculated from the cycle (sense of machining).
- At the end of the groove, the tool is retracted at an angle of 45° and 0.5 mm away from the groove side

Remark:

Make sure that the tool orientation physically corresponds with the actual tool position: Left/Right or In/Outside cutting edge.

19.13.16 G849 Grooving universal radial, finishing



```

G   Grooving radial universal finish
Y   Starting point
Z   Starting point
Y1= Beginpoint contour
Z1= Beginpoint contour
Y2= Groove depth
R2= Radius 2
Y3= Endpoint contour
Z3= Endpoint contour
A   Angle 1
B   Angle 2
I1= Chamfer length 1
I3= Chamfer length 3
R1= Radius 1
R3= Radius 3

```

```

S1= (Cutting) Speed
F   Feed

```

Y, Z Starting point grooving cycle.
 Y1=, Z1= Contour Starting point
 Y2= Contour bottom
 Y3=, Z3= Contour end point. If Y3 is not programmed then (Y3=Y1)
 A Angle (0-89°) at groove starting point (Y1, Z1)
 B Angle (0-89°) at groove end point (Y3, Z3)
 I1= Chamfer length at groove starting point (Y1, Z1)
 I3= Chamfer at groove end point (Y3, Z3)
 R1= Rounding at groove starting point. (Y1, Z1)
 R2= Rounding at both sides of groove bottom.
 R3= Rounding at groove end point (Y3, Z3)

Basic settings: A=0, B= 0, I1=0, R1= 0, I3=0, R3=0, R2=0

Associated functions: G845 for roughing

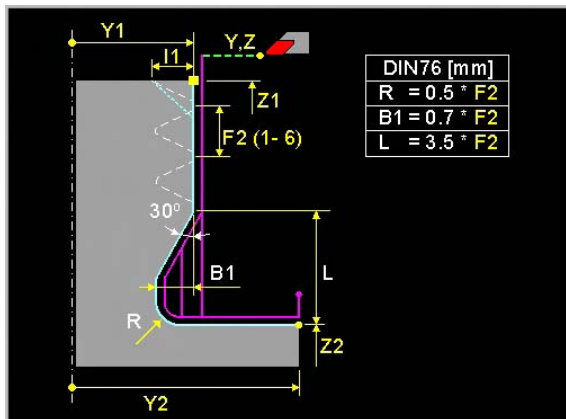
Notes and application

- First the opposite groove side is cut, than the adjoining groove side followed by the groove bottom
- The tool width (C6) is taken from the tool table. An error code appears if the tool width is not available.
- Tool orientation (O):
 - . The tool orientation is stored in the tool table
 - . With the G-function G302, the tool orientation can be overwritten in the program
 - . If there is no tool orientation available, the tool orientation will be calculated from the cycle (sense of machining).
- At the end of the groove, the tool is retracted at an angle of 45° and 0.5 mm away from the groove side

Remark:

Make sure that the tool orientation physically corresponds with the actual tool position: Left/Right or In/Outside cutting edge.

19.13.17 G850 Undercut DIN76



```

G   Undercut (DIN 76)
Y   Starting point
Z   Starting point
Y1= Beginpoint contour
Z1= Beginpoint contour
Y2= Endpoint contour
Z2= Endpoint contour
F2= Pitch
I1= Chamfer length 1
S1= (Cutting) Speed
F   Feed
  
```

Y, Z Starting point undercut cycle.
 Y1=, Z1= Contour starting point
 Y2=, Z2= Contour endpoint.
 F2= Pitch (1-6)
 I1= Chamfer length

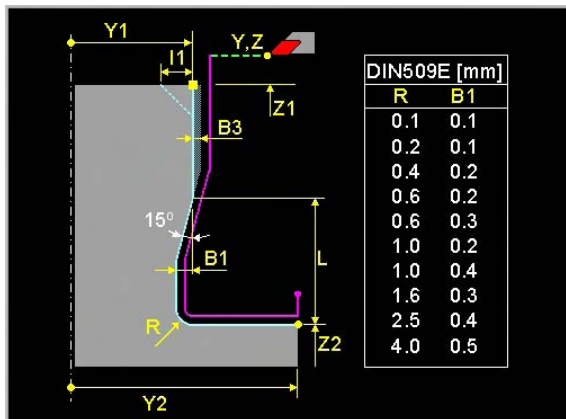
Basic settings

I1=0

Notes and application

- The undercut contour consists of the elements: Chamfer (Optional), Cylinder, Undercut geometry, Face surface on a pre-cut contour shape.
- Only undercuts conform the DIN-norm can be programmed.
- Undercut (DIN-norm):
 - . Length is $F2 \times 3.5$
 - . Depth is $F2 \times 0.7$
 - . Radius is $F2 \times 0.5$
 - . Angle is 30° fixed
- Sequence:
 - Start motion axis parallel from starting point (Y, Z) to contour starting point (Y1=, Z1=)
 - Roughing movement of the undercut shape to contour endpoint (Y2=, Z2=).
Depending on the pitch (F2), the undercut shape will be cut in multiple cuts.
 - Finishing of the complete undercut shape
 - At the contour endpoint, the Z-axis retracts 0.1 mm from the contour

19.13.18 G851 Undercut DIN 509 E



G Undercut (DIN 509 E)
 Y Starting point
 Z Starting point
 Y1= Beginpoint contour
 Z1= Beginpoint contour
 Y2= Endpoint contour
 Z2= Endpoint contour
 R Radius
 B1= Depth of undercut
 L Length of undercut
 B3= Grinding allowance
 I1= Chamfer length 1
 S1= (Cutting) Speed
 F Feed

Y, Z Starting point undercut cycle.
 Y1=, Z1= Contour starting point
 Y2=, Z2= Contour endpoint
 R Radius of the undercut shape
 B1= Undercut depth
 L Undercut length
 B3= Finishing allowance.
 I1= Chamfer length

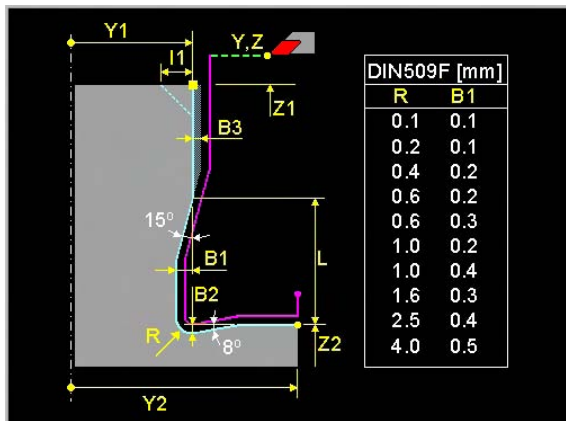
Basic settings

I1=0

Notes and application

- The undercut contour consists of the elements: Chamfer (Optional), Cylinder, Undercut geometry, Face surface on a pre-cut contour shape.
- Undercuts can be programmed conform the DIN-norm or free-form
 - . For DIN-norm undercut values for depth (B1) and radius (R) can be taken from the table.
 - . For free form undercuts (B1) and (R) are free programmable
- Sequence:
 - Start motion axis parallel from starting point (Y, Z) to contour starting point (Y1=, Z1=)
 - Roughing movement of the undercut shape to contour endpoint (Y2=, Z2=).
 - Finishing of the complete undercut shape
 - At the contour endpoint, the Z-axis retracts 0.1 mm from the contour

19.13.19 G852 Undercut DIN 509 F



G Undercut (DIN 509 F)
 Y Starting point
 Z Starting point
 Y1= Beginpoint contour
 Z1= Beginpoint contour
 Y2= Endpoint contour
 Z2= Endpoint contour
 R Radius
 B1= Depth of undercut
 L Length of undercut
 B2= Depth of undercut
 B3= Grinding allowance
 I1= Chamfer length 1
 S1= (Cutting) Speed
 F Feed

Y, Z Starting point undercut cycle.
 Y1=, Z1= Contour starting point
 Y2=, Z2= Contour endpoint
 R Radius of the undercut shape
 B1= Undercut depth
 L Undercut length
 B2= Undercut depth
 B3= Finishing allowance.
 I1= Chamfer length

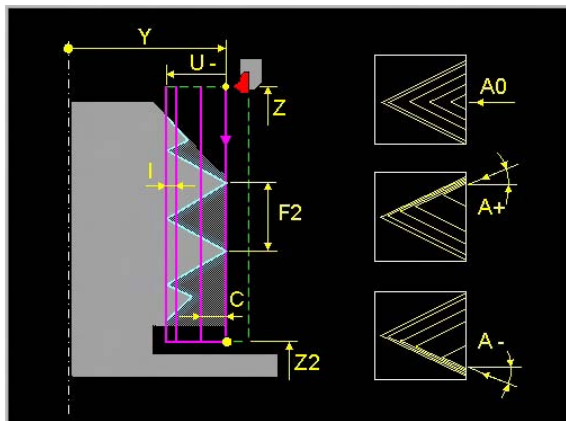
Basic settings

I1=0

Notes and application

- The undercut contour consists of the elements: Chamfer (Optional), Cylinder, Undercut geometry, Face surface on a pre-cut contour shape.
- Undercuts can be programmed conform the DIN-norm or free-form
 - . For DIN-norm undercut values for depth (B1) and radius (R) can be taken from the table.
 - . For free form undercuts (B1) and (R) are free programmable
- Sequence:
 - Start motion axis parallel from starting point (Y, Z) to contour starting point (Y1=, Z1=)
 - Roughing movement of the undercut shape to contour endpoint (Y2=, Z2=).
 - Finishing of the complete undercut shape
 - At the contour endpoint, the Z-axis retracts 0.1 mm from the contour

19.13.20 G861 Threading axial



```

G   Threadcutting axial
Y   Starting point
Z   Starting point
Z2= Endpoint thread
C   Cutting depth
U   Thread depth
A   In-feed angle
I   Cutting depth last pass
K1= Number of multiple threads
F2= Pitch
I1= 0=cut segmentation 1=single cut
S1= Speed

```

Y, Z Starting point threading cycle.
 Z2= End point. At the end point the Y-axis will be retracted at an angle of 90° to (Y) and the Z-axis moves in rapid traverse back to (Z)
 C In-feed depth is calculated from: in-feed angle (A), threading depth (U) and finishing allowance (I). Minimum in-feed depth: 0.002
 U Threading depth (+/- U) is calculated from pitch (F2):
 Outside thread $U = -0.6495 \times F2$; Inside thread $U = 0.6403 \times F2$
 U -999: Outside thread with calculation (Default)
 U 999: Inside thread with calculation
 A In-feed angle (Default 28°)
 $A = -45^\circ < A < 45^\circ$; In-feed along the thread edge.
 $A = 0^\circ$; In-feed only in Y-direction
 I Last cut at thread depth. Minimum value (Default): 0.010
 K1= Number of thread cuts. (Default 1). $1 < K1 = < 99$
 F2= Pitch in mm/revolution.
 I1= Single cut. The thread will be cut in one pass to depth. (Thread finish)
 S1= Spindle revolution Rev./Min (G97)

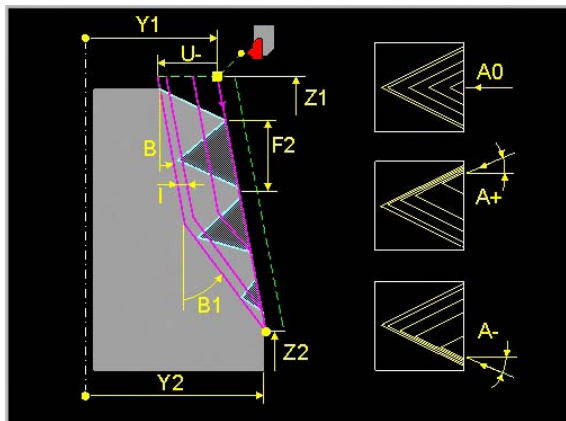
Basic settings

U=+ 999, A=28°, I=0.010, K1=1, I1=0,

Notes and application

- The turning table should be programmed in revolution/min (S1) (G97).
- Regard the maximum feed (feed is $S1 = \times F2$).
- Feed and spindle revolution override is not operational during thread cutting
- The turning table speed is fixed during thread cutting.
- Threading can be interrupted but stops only at the end of the thread cut.
- Regard the turning table direction (M1=03/04) and the tool orientation (O)

19.13.21 G862 Treading conical



G Treadcutting taper
 Y Starting point
 Z Starting point
 Y1= Beginpoint thread
 Z1= Beginpoint thread
 Y2= Endpoint thread
 Z2= Endpoint thread
 C Cutting depth
 U Thread depth
 A In-feed angle
 B Taper angle
 B1= Withdrawal angle
 I Cutting depth last pass
 K1= Number of multiple threads
 F2= Pitch

I1= 0=cut segmentation 1=single cut
 S1= Speed

- Y, Z Starting point threading cycle.
 Z2= End point. At the end point the Y-axis will be retracted at an angle of 90° to (Y) and the Z-axis moves in rapid traverse back to (Z)
 C In-feed depth is calculated from: in-feed angle (A), threading depth (U) and finishing allowance (I). Minimum in-feed depth: 0.002
 U Threading depth (+/- U) is calculated from pitch (F2):
 Outside thread $U = -0.6495 \times F2$; Inside thread $U = 0.6403 \times F2$
 U -999: Outside thread with calculation (Default)
 U 999: Inside thread with calculation
 A In-feed angle (Default 28°)
 $A = -45^\circ < A < 45^\circ$; In-feed along the thread edge.
 $A = 0^\circ$; In-feed only in Y-direction
 I Last cut at thread depth. Minimum value (Default): 0.010
 K1= Number of thread cuts. (Default 1). $1 < K1 = < 99$
 F2= Pitch in mm/revolution.
 I1= Single cut. The thread will be cut in one pass to depth. (Thread finish)
 S1= Spindle revolution Rev./Min (G97)
 B Cone angle in relation with the Z-axis ($-45^\circ < B < 45^\circ$). (B/Y1=) or B/Y2=) has to be programmed.
 B1= Run-out angle at the end of thread (Default 45°) ($0^\circ < B1 = < 90^\circ$)
 I Last cut at thread depth. Minimum value (Default): 0.010
 K1= Number of thread cuts. (Default 1). $1 < K1 = < 99$
 F2= Pitch in mm/revolution.
 I1= Single cut. The thread will be cut in one pass to depth. (Thread finish)
 S1= Spindle revolution Rev./Min (G97)

Basic settings

U=+ 999, A=28°, I=0.010, K1=1, I1=0,

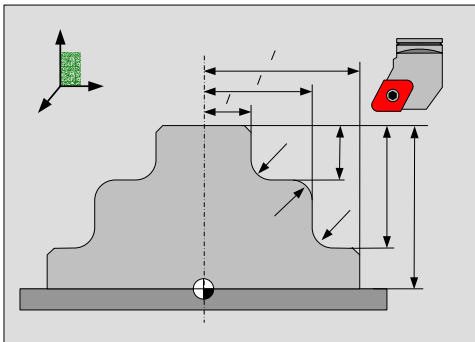
Notes and application (see G861)

19.14 Examples

Example 1

Program	Description
N9999	
N1 G17	Set planes for milling. Length compensation in Z direction.
N2 G37	Milling mode
N3 M54	Head is in the Z direction
N4 T1 M6	Insert milling tool
N5 S1000 F1000 M3	Start Spindle
N...	Milling
N100 G17 Z1=1 Y1=2	Set planes for turning. Main axis 1 is Z, main axis 2 is Y. Radius correction in ZY plane.
N101 G36	Turning
N102 T7 M6	Insert tool
N103 S1=100 M1=3	Start rotary table for continuous turning
N104 G0 X0 Y100 Z100	Position turning tool
N105 G822	Start longitudinal cutting cycles
N...	Turning
N200 G17	Set planes for milling. Length compensation in Z direction.
N201 G37	Milling mode
N203 T1 M6	Insert milling tool
N204 S1000 M3	Start Spindle
N205	Position milling tool
N300 M30	Program end

Example 2: Workpiece drawing Example 2:



Program	Description
N9999	
N1 G17	Set planes for milling. Length compensation in Z direction
N2 G37	Milling mode
N3 G54 I1 Z8	Zero point displacement for Z direction. Upper edge of material is zero
N4 G36	Turning
N5 M54	Head is in the Z direction
N6 G17 Z1=1 Y1=2	Set planes for turning. Main axis 1 is Z, main axis 2 is Y. Radius correction in ZY plane
N7 G195 X-1 Y-1 Z1 I2 J12 K-11.	Set graphics window
N8 G199 X0 Y0 Z0 B4 C2	Start of material graphical contour description. B4 means automatic drawing.
N9 G198 I1=14 X0 Y8 Z0	Start of contour description. I1=14 is light blue colour
N10 G2 X0 Y8 I0 J0	Upper circle of cylinder
N11 G1 X0 Y8 Z-8	Line
N12 G2 X0 Y8 I0 J0	Lower circle of cylinder
N13	End of graphical contour description
N14 T1 M6 (L100 R5 C0.3 Q3=800)	Insert turning tool (length, radius, corner radius and type)
N15 S1=1000 M1=3	Start rotary table for continuous turning
N16 G0 X0 Y8 Z3 F1000	Position turning tool
N17	
N18 G823 Y8 Z0.3 Y1=8 Z1=-3 Y2=2 Z2=0 I1=0.5 R2=0.5 C0.2	G823 start cutting plan cycles. Turn upper part
N19 G823 Y8 Z-2.7 Y1=8 Z1=-6 Y2=5 Z2=-3 R1=0.5 I2=0.5 R2=0.5 C0.2	G823 start cutting plan cycles. Turn lower part
N20	
N21 G827 Y8 Z-6.7 Y1=8 Z1=-6 Y2=5 Z2=-3 R1=0.5 I2=0.5 R2=0.5	G827 start finish machining cutting plan cycles. Finish machine lower part
N22 G827 Y8 Z-2.7 Y1=8 Z1=-3 Y2=2 Z2=0 I1=0.5 R2=0.5	G827 start finish machining cutting plan cycles. Finish machine upper part
N23 G0 Z10	Move tool clear
N24 T0 M6	Reset tool
N25 G37	Milling mode
N26 G53	Deactivate zero point displacement
N300 M30	Program end

19.15 Survey of permitted G-Functions in the turning mode.

The permitted G-Functions applicable in the turning mode are listed in the tabel underneath. For more information about the G-Functions refer to the control system user manual.

G-Funkions in Turning mode	Explanation
G00	Rapid traverse
G01	Linear interpolation
G02/G03	Circular clockwise/Circular counter clockwise
G04	Dwell time
G14	Repeat function
G17/G18	Main plane
G22	Macro call
G23	Main program call
G25/G26	Enable/Disable feed and spindel override
G27/G28	Reset/Activate positioning functions
G29	Conditional jump
G33	Basic threatcutting movement
G36/G37	Switching turning mode on and off
G39	Activate/Deactivate offset
G40-G41/G42,G43/G44	Tool radius compensation
G45- -50	Measuring cycles
G53/G54- -G59	Cancel/Activate zero point shift
G63/G64	Cancel/Activate geometric calculations
G70/G71	Inch/Metric Programming
G90/G91	Absolute/Incremental programming
G92/G93	Zeropoint shift incremental/absolute
G94/G95	Feed in mm/min or mm/rev
G96/G97	Constant cutting speed
G98/G99, G195, G196, G197/G198, G199	Graphic functions
G227/G228	Unbalance monitor
G300- -G351	Special functions for macros
G611- -G615	Measuring cycles
G691/G692	Unbalance cycles
G822- -G823- -G826- -G827	Clearance cycles
G832- -G833- -G836- -G837	Roughing cycles
G842- -G843- -G846- -G847	Grooving cycles

20. G64 Geometric calculations with continuous movements

20.1 Conventions with the formats

For all the formats in this appendix the G64-function is assumed to have already been programmed in a previous block and is therefore active.

The XY-plane is also assumed to be the active plane; if another plane is active the appropriate addresses must be substituted in the formats.

To show that more than one block is required with a particular format, the first block is numbered as N1 and the following as N2, N3 etc. The use of these numbers is not compulsory; they have been used purely as a convention.

In the formats, programming the end point is indicated with **X.. Y...** but instead of these coordinates the polar coordinates B2=, L2=.. or a defined point P.. can be used too.

Sometimes, programming the endpoint is indicated with [endpoint]. In this case the endpoint can be programmed as outside the geometry (G63 active). Thus with:

X.. only, or Y.. only, or X.. and Y.. or B1=.. and X.. or B1=.. and Y..

In the formats, programming the centre point of a circle is indicated with I.. J... but instead of these coordinates the polar coordinates B3=, L3=.. can be used.

The use of a support point is indicated with **X.. Y.. I1=0** and of a parallel line with X.. Y.. I1= ... Instead of X.. and Y.. the polar coordinates B2=, L2=.. or a defined point P.. can be used too. It is also possible to use X1=.. Y1=.. Instead of X.. Y.. I1=0 In some cases a support point or a parallel line can be used. This is indicated with X.. Y.. I1=0 or I1=+...

Sometimes a support point is indicated with [support point]. All possible formats for support point can be used.

In the illustrations in which the formats are explained, the following conventions are used:

P_o = a start point known from the previous block

P_s = a support point on a line or on a parallel line

P_e = a programmed end point

M = a programmed circle centre point

R = a programmed radius of a circle

A lot of line definitions are given with an angle B1=.. or a support point P, on the line. This is indicated in the illustrations with {B1=} and {Ps}. If B1= and Ps are drawn without the {} both words have to be programmed.

Contents of this section

Intersection point

- 1.1 Intersection point of two straight lines
- 1.2 Intersection point of two lines programmed as end point
- 1.3 Chamfer between intersecting straight lines
- 1.4 Rounding between intersecting straight lines
- 1.5 Rounding between straight line and chamfer
- 1.6 88888intersection point indicator (j1=)
- 1.6 Intersection point between line and circle
- 1.7 Intersection point of line and circle programmed as end point
- 1.8 Rounding between intersecting line and circle
- 1.9 Intersection point between circle and line
- 1.10 Intersection point of circle and line programmed as end point
- 1.11 Rounding between intersecting circle and line
- 1.12 Intersection point between two circles
- 1.13 Intersection point of two circles programmed as end point
- 1.14 Rounding between two intersecting circles

Point of tangency

- 2.1 Point of tangency indicator (r1=)
- 2.2 Tangent line and circle
- 2.3 Connecting circle between tangent line and circle
- 2.4 Tangent circle and line
- 2.5 Connecting circle between tangent circle and line
- 2.6 Two tangent circles
- 2.7 Continuous connecting circle between two tangent circles

Connecting circle between elements, which do not meet

- 3.1 Line and circle
- 3.2 Circle and line
- 3.3 Two circles outside each other
- 3.4 One circle inside the other one
- 3.5 Concentric circles

Geometric calculations with non-continuous movements G64

- 4.1 Rounding or connecting circle indicator (k1=)
- 4.2 Rounding with intersection points
- 4.3 Rounding between intersecting straight lines
- 4.4 Rounding between intersecting line and circle
- 4.5 Rounding between intersecting circle and line
- 4.6 Rounding between two intersecting circles
- 4.7 Tangent lines (r1=)
- 4.8 Connecting circle between tangent line and circle or v.v.
- 4.9 Connecting circle between a line which do not meet a circle
- 4.10 Connecting circle between circles outside each other
- 4.11 Connecting circle between two circles one inside the other
- 4.12 Connecting circle with two concentric circles

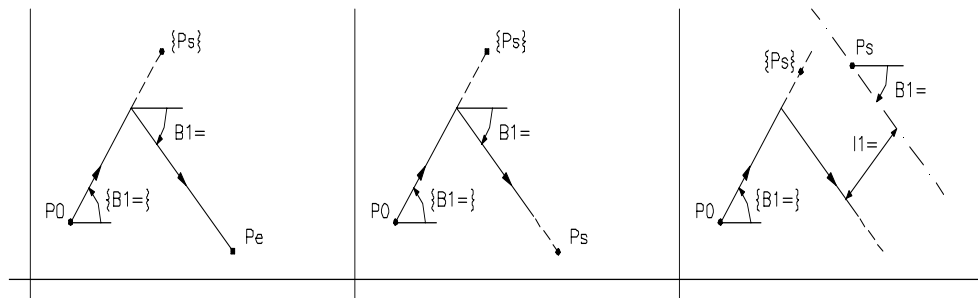
20.2 Intersection point

20.2.1 Intersection point of two straight lines

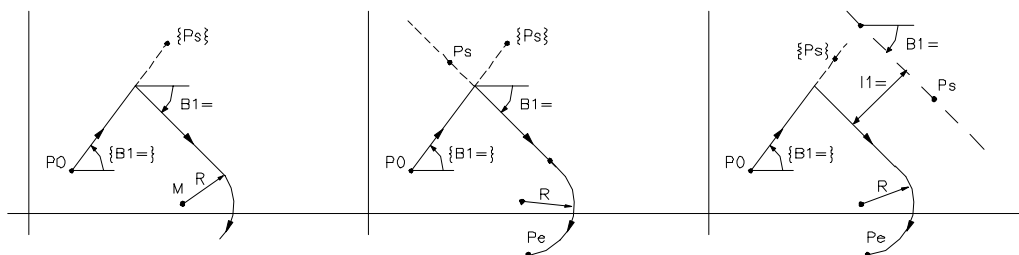
To calculate the intersection point between two lines

Start point from N1 is known

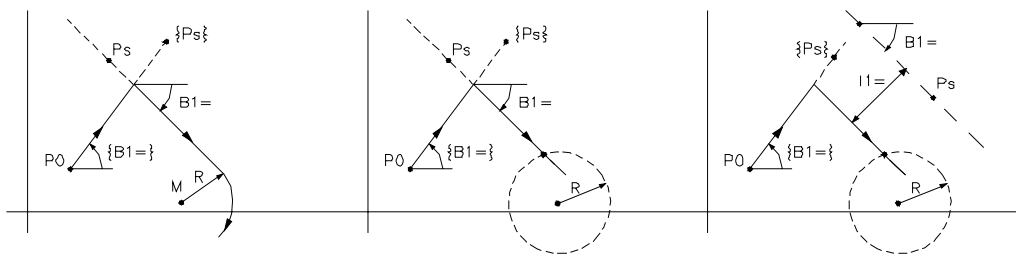
If the start point of the first line is known either the angle with the main axis or any support point on the line can be used to define the first line. Several formats are possible for the second line.



N1	G1	B1=..			
N2		B1=..	X..	Y..	
or					
N1	G1	B1=..			
N2		B1=..	X..	Y..	I1=0 or I1=±..
or					
N1	G1	X..	Y..	I1=0	
N2		B1=..	X..	Y..	
or					
N1	G1	X..	Y..	I1=0	
N2		B1=..	X..	Y..	I1=0 or I1=±..



N1	G1	B1=..			
N2		B1=..	R1=0		
or					
N1	G1	B1=..			
N2		B1=..	X..	Y..	I1=0 or I1=±.. R1=0
or					
N1	G1	X..	Y..	I1=0	
N2		B1=..	R1=0		
or					
N1	G1	X..	Y..	I1=0	
N2		B1=..	X..	Y..	I1=0 R1=0



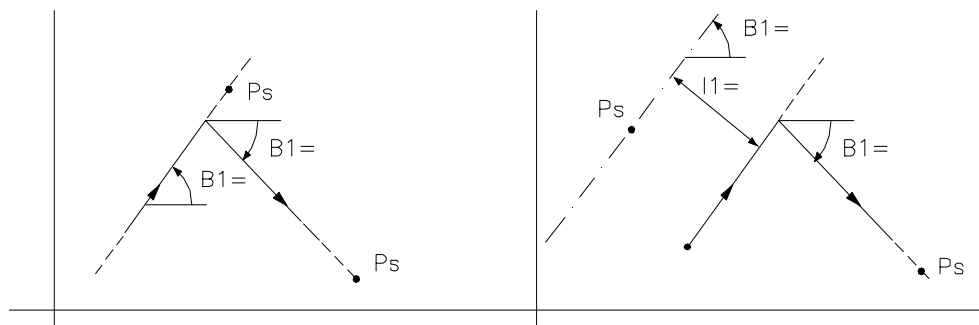
```

N1      G1      B1=..
N2      X..     Y..     I1=0    R1=0
or
N1      G1      B1=..
N2      B1=..   X..     Y..     I1=0 or I1=..    J1=1/2
or
N1      G1      X..     Y..     I1=0
N2      X..     Y..     I1=0    R1=0
or
N1      G1      X..     Y..     I1=0
N2      B1=..   X..     Y..     I1=0 or I1=..    J1=1/2

```

Start point from N1 is not known

If the start point from N1 is not known, both the angle and a support point have to be programmed in block N1. So this block reads:



```

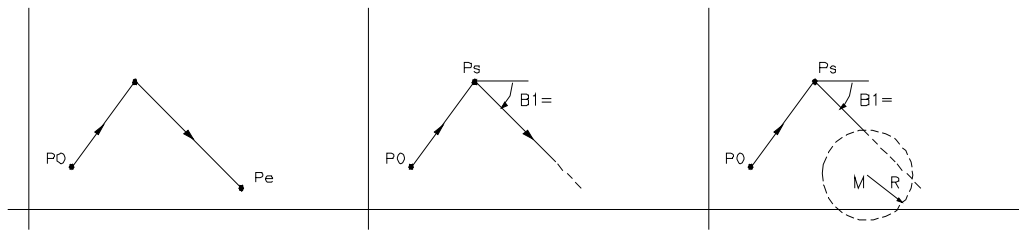
N1      G1      B1=..   X..     Y..     I1=0 or I1=..

```

Block N2 from the mentioned cases remains the same.

support point coincides with the point of intersection

If the support point coincides with the point of intersection, it is assumed that this point is the start point of the next line. This results in a few additional formats in which the second line can be programmed with either the angle or a support point on the line:

Start point from N1 is known

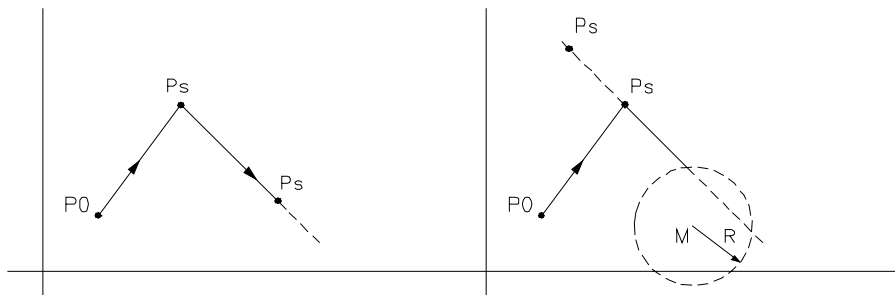
N1 G1 [support point = Intersection point]
 N2 X.. Y..

or

N1 G1 [support point = Intersection point]
 N2 B1=..

or

N1 G1 [support point = Intersection point]
 N2 B1=.. J1=1/2



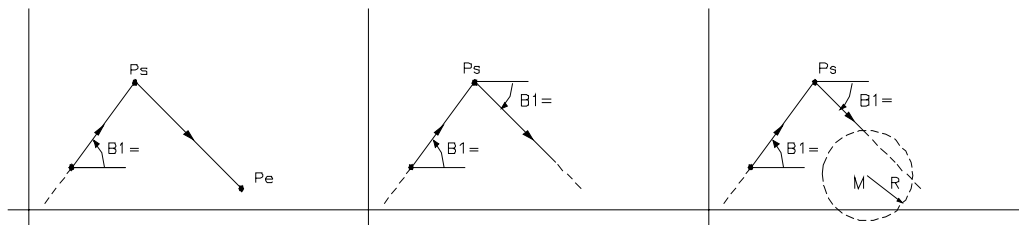
N1 G1 [support point = Intersection point]
 N2 X.. Y.. I1=0

or

N1 G1 [support point = Intersection point]
 N2 X.. Y.. I1=0 J1=1/2

Start point from N1 is not known

If the start point from N1 is not known, both the angle and the support point have to be programmed in block N1. So this block reads::



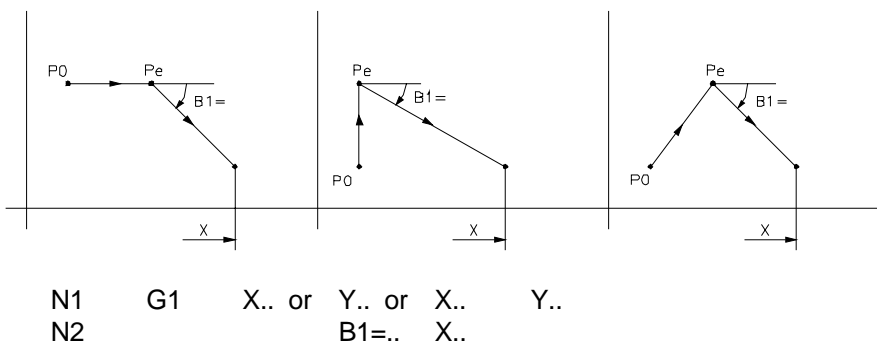
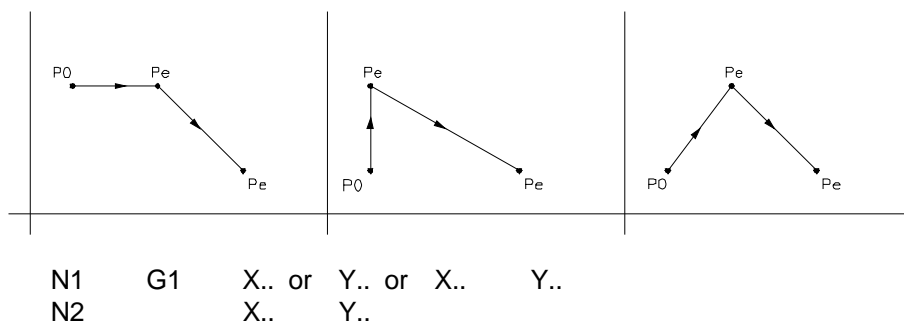
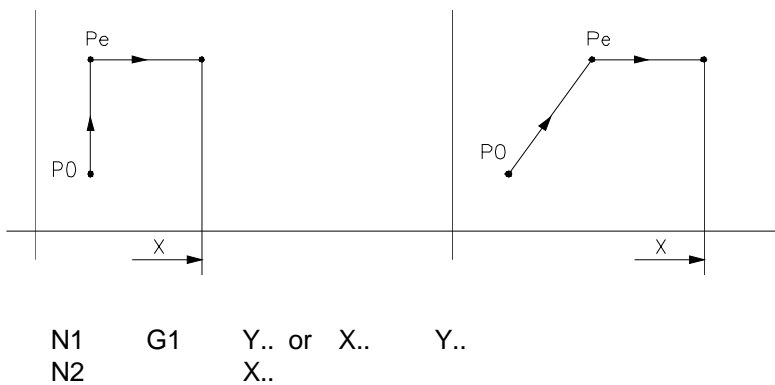
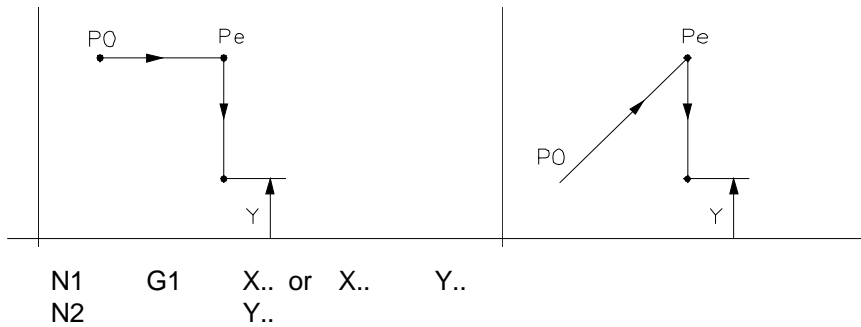
N1 G1 B1=.. [support point = Intersection point]

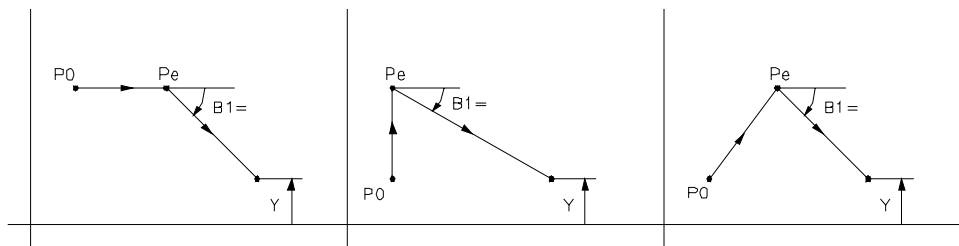
Block N2 from the mentioned cases remains the same.

20.2.2 Intersection point programmed as end point

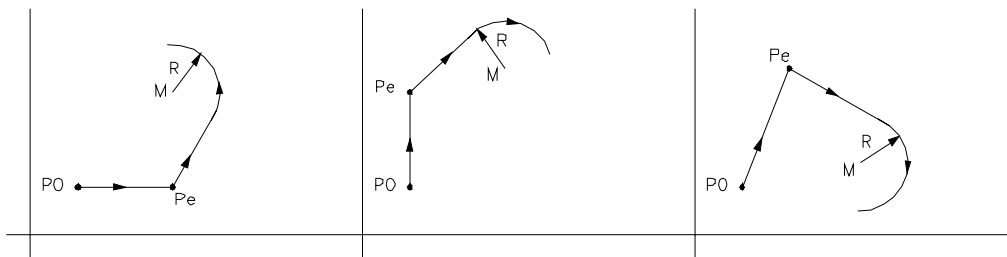
In some cases the intersection point of two lines is known from the drawing and can be programmed directly. It is assumed that this point is the start point of the next line. If the end point is programmed with one coordinate only, the other coordinate is picked up from the previous blocks. The following extra formats are possible:

Start point from N1 is known





N1	G1	X..	or	Y..	or	X..	Y..
N2				B1=..		Y..	

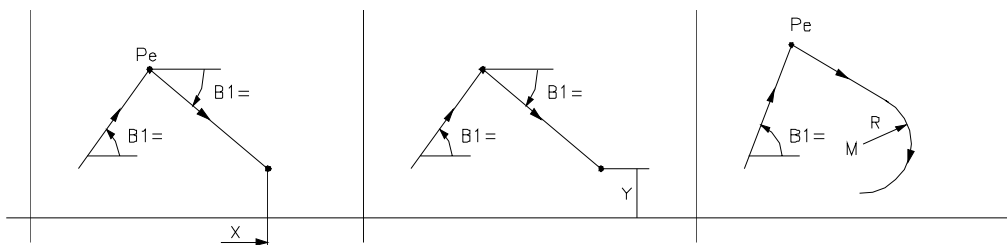
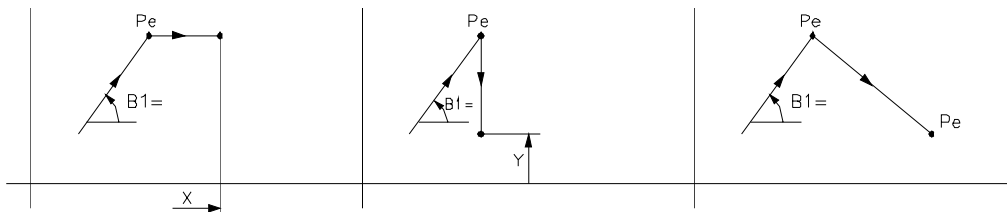


N1	G1	X..	or	Y..	or	X..	Y..
N2				R1=0			

All formats from the previous case for calculating the intersection point can be programmed too.

Start point from N1 is not known

If the start point from N1 is not known, both the angle and the endpoint have to be programmed in block N1. So this block reads:



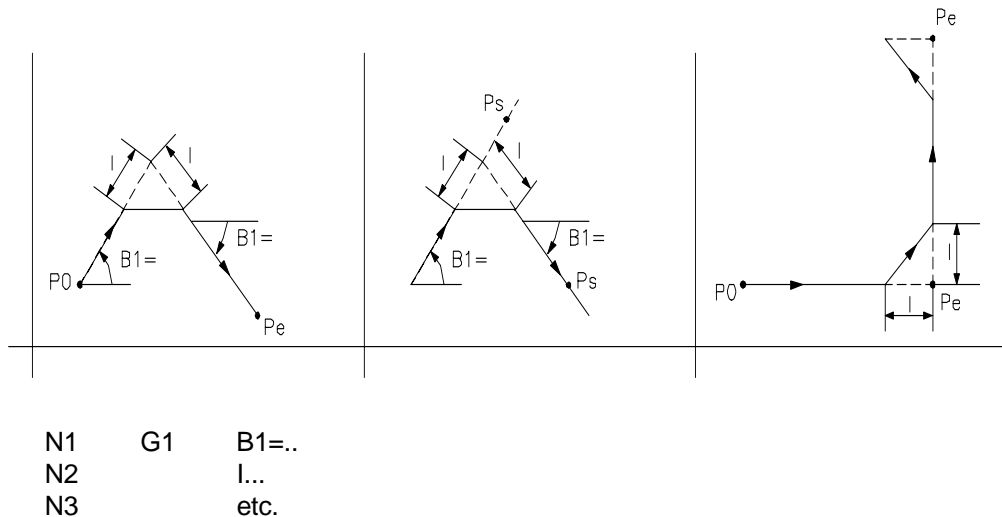
N1	G1	B1=.	X..	
or				
N1	G1	B1=.	Y..	
or				
N1	G1	B1=.	X..	Y..

Block N2 from the mentioned cases remains the same.

20.2.3 Chamfer between intersecting straight lines

To insert a symmetrical chamfer between two straight lines

Start point from N1 is known



Refer to block N2 of the previous sections for the formats of block N3.

Note: instead of programming B1=.. in block N1 it is also possible to use a support point, a parallel line or an end point with either X.. or Y.. or X.. and Y..

start point from N1 is not known

If the start point from N1 is not known, both the angle and a support point or end point have to be programmed in block N1. So this block reads:

```

N1 G1 B1=.. X.. Y.. l1=0 or l1=..
N2  l...
N3  etc.
```

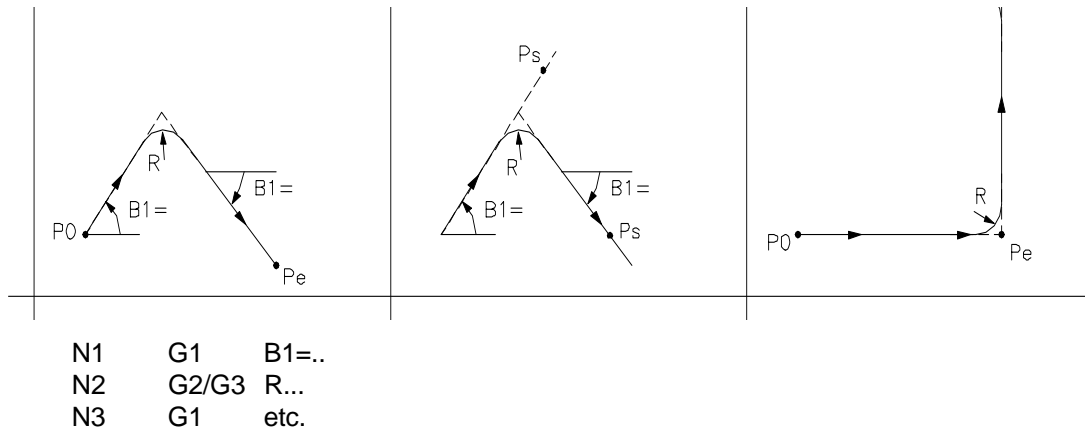
The blocks N2 and N3 are the same as with a known start point. So refer to that section for these blocks.

Note: instead of programming B1=.. with a support point or parallel line in block N1 it is also possible to use an end point with either X.. or K. or X.. and Y..

20.2.4 Rounding between intersecting straight lines

To insert a rounding between two straight lines

Start point from N1 is known



Note: instead of programming B1=.. in block N1 it is also possible to use a support point, a parallel line or an end point with either X.. or Y.. or X.. and Y..

Refer to block N2 of the previous sections for the formats of block N3.

start point from N1 is not known

If the start point from N1 is not known, both the angle and a support point or end point have to be programmed in block N1. So this block reads:

```

N1 G1 B1=.. X.. Y.. I1=0 or I1=∇..
N2 G2/G3 R...
N3 G1 etc.
```

Note: instead of programming B1=.. with a support point or parallel line in block N1 it is also possible to use an end point with either X.. or Y.. or X.. and Y..

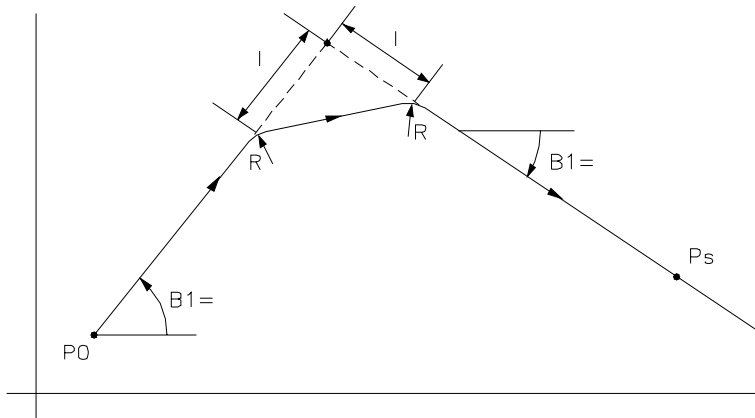
The blocks N2 and N3 are the same as with a known start point. So refer to that section for these blocks.

20.2.5 Rounding between straight line and chamfer

To insert a rounding between a straight line and a chamfer

In the following formats both roundings are indicated. It is possible to insert just one rounding and leave out the other one.

Start point from N1 is known



N1	G1	B1=..
N2	G2/G3	R...
N3	G1	I...
N4	G2/G3	R...
N5	G1	etc.

Note: instead of programming B1=.. in block N1 it is also possible to use a support point, a parallel line or an end point with either X.. or Y.. or X.. and Y..

Refer to block N2 of the previous sections for the formats of block N5.

start point from N1 is not known

If the start point from N1 is not known, both the angle and a support point or end point have to be programmed in block N1. So this block reads:

```

N1 G1 B1=.. X.. Y.. I1=0 or I1=∇..
N2 G2/G3 R...
N3 G1 I...
N4 G2/G3 R...
N5 G1 etc.
```

Note: instead of programming B1=.. with a support point or parallel line in block N1 it is also possible to use an end point with either X.. or Y.. or X.. and Y..

Refer to block N2 of the previous sections for the formats of block N5.

20.2.6 Intersecting point between line circle

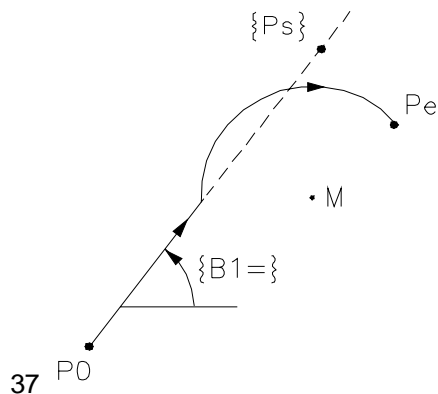
To calculate the point of intersection between line and circle

Intersecting point indicator (J1=)

Refer to the description of the INTERSECTION POINT INDICATOR in the Notes and Usage of the function G64.

Start point from N1 is known

If the start point of the line is known either the angle the line makes with the main axis or any support point on the line can be used to define the line. Several formats are possible for the circle:

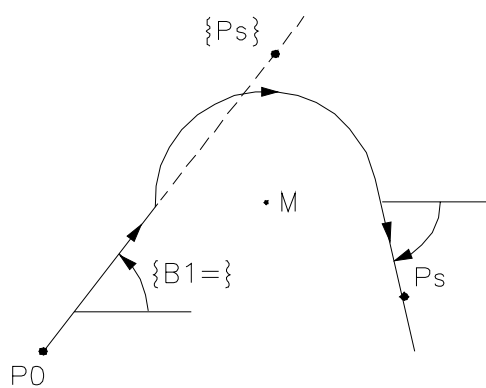
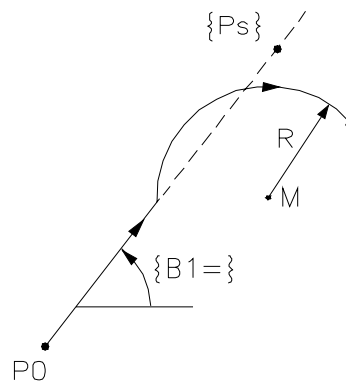


N1	G1	B1=.	J1=1/2		
N2	G2/G3	I..	J..	X..	Y..

or					
N1	G1	B1=.	J1=1/2		
N2	G2/G3	I..	J..	R..	

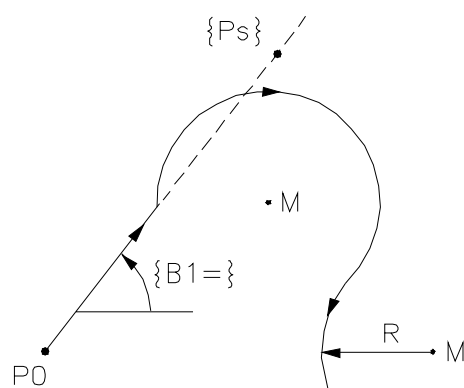
or					
N1	G1	X..	Y..	I1=0	J1=1/2
N2	G2/G3	I..	J..	X..	Y..

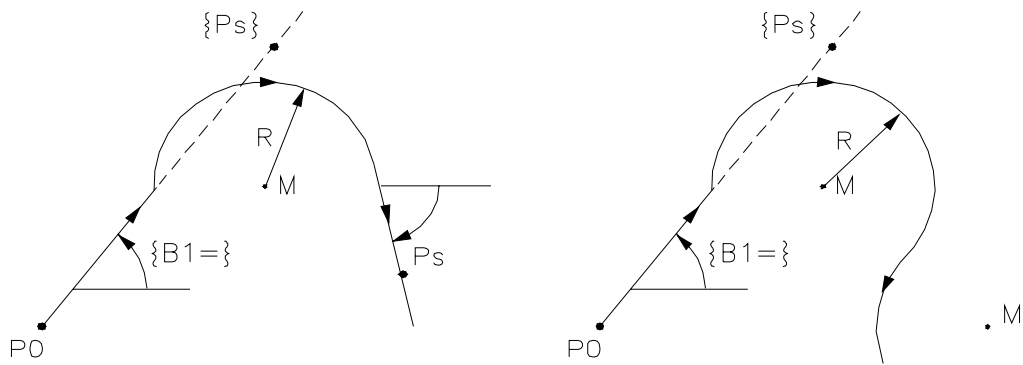
or					
N1	G1	X..	Y..	I1=0	J1=1/2
N2	G2/G3	I..	J..	R..	



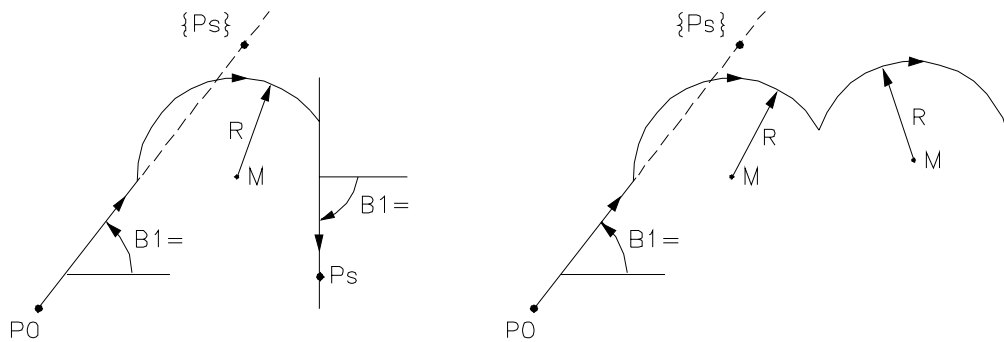
N1	G1	B1=.	J1=1/2		
N2	G2/G3	I..	J..	R1=0	

or					
N1	G1	X..	Y..	I1=0	J1=1/2
N2	G2/G3	I..	J..	R1=0	





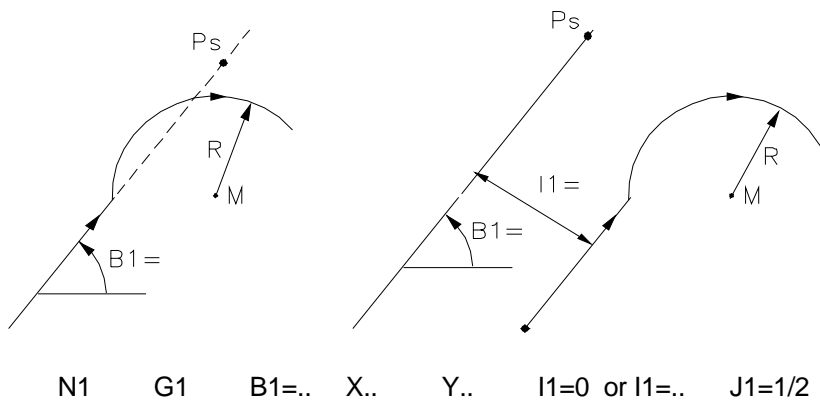
N1	G1	B1=..	J1=1/2		
N2	G2/G3	I..	J..	R..	R1=0
or					
N1	G1	X..	Y..	I1=0	J1=1/2
N2	G2/G3	I..	J..	R..	R1=0



N1	G1	B1=..	J1=1/2		
N2	G2/G3	I..	J..	R..	J1=1/2
or					
N1	G1	X..	Y..	I1=0	J1=1/2
N2	G2/G3	I..	J..	R..	J1=1/2

Start point from N1 is not known

If the start point from N1 is not known, both the angle and a support point have to be programmed in block N1. So this block reads:



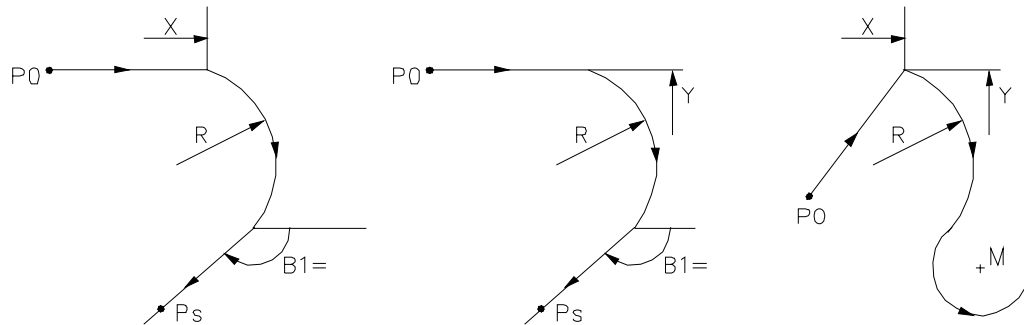
N1	G1	B1=..	X..	Y..	I1=0 or I1=..	J1=1/2
----	----	-------	-----	-----	---------------	--------

Block N2 from the mentioned cases remains the same.

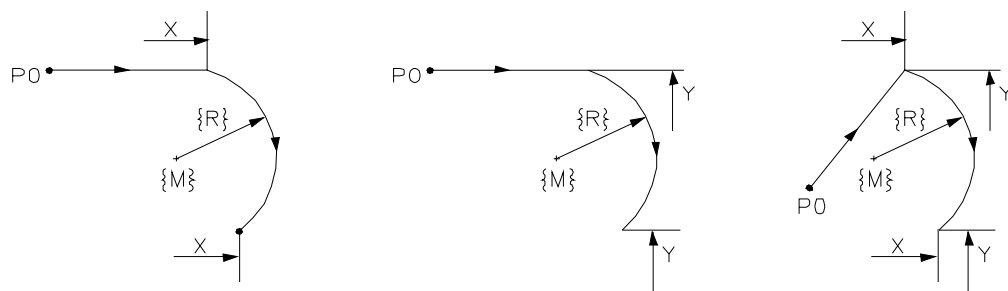
20.2.7 Intersecting point of line and circle programmed as end point

In some cases the intersection point of the line and circle is known from the drawing and can be programmed directly. It is assumed that this point is the start point of the next movement. The end point can be programmed with one or two coordinates and if the start point of the line is not known, the angle, which the line makes with the main axis, can be added to the block. The following formats are possible:

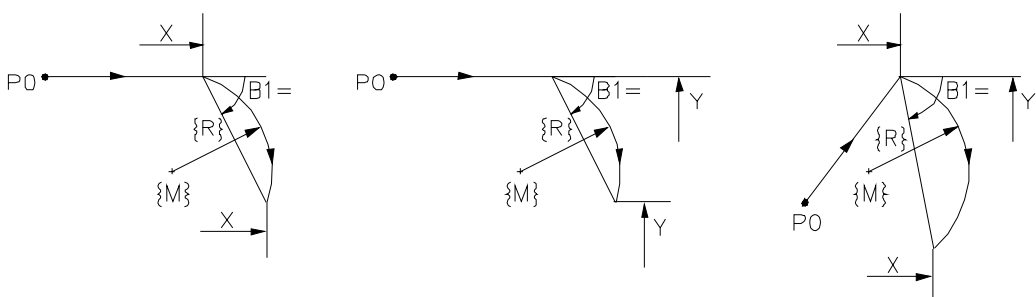
Start point from N1 is known



N1	G1	X.. or	Y.. or	X..	Y..
N2	G2/G3	R..	R1=0		

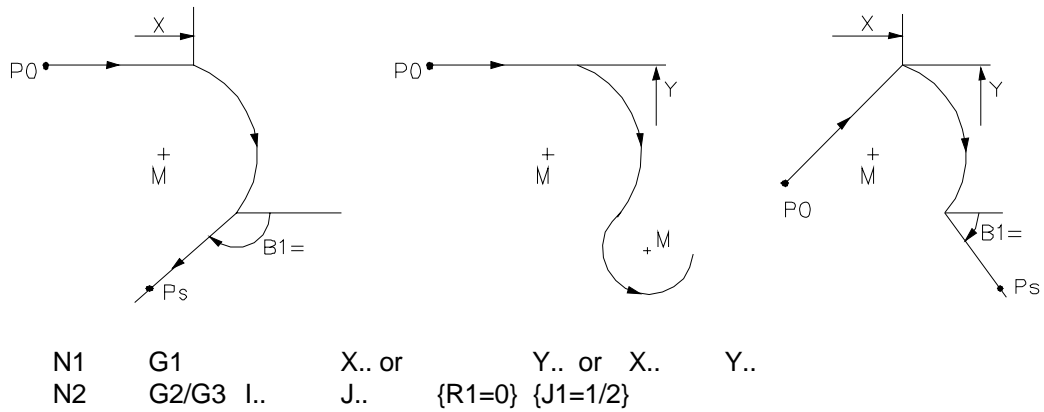


N1	G1	X.. or	Y.. or	X..	Y..
N2	G2/G3	R..	X.. or	Y.. or	X..
					Y..
or					
N1	G1	X.. or	Y.. or	X..	Y..
N2	G2/G3	I..	J..	X.. or	Y.. or X..Y..



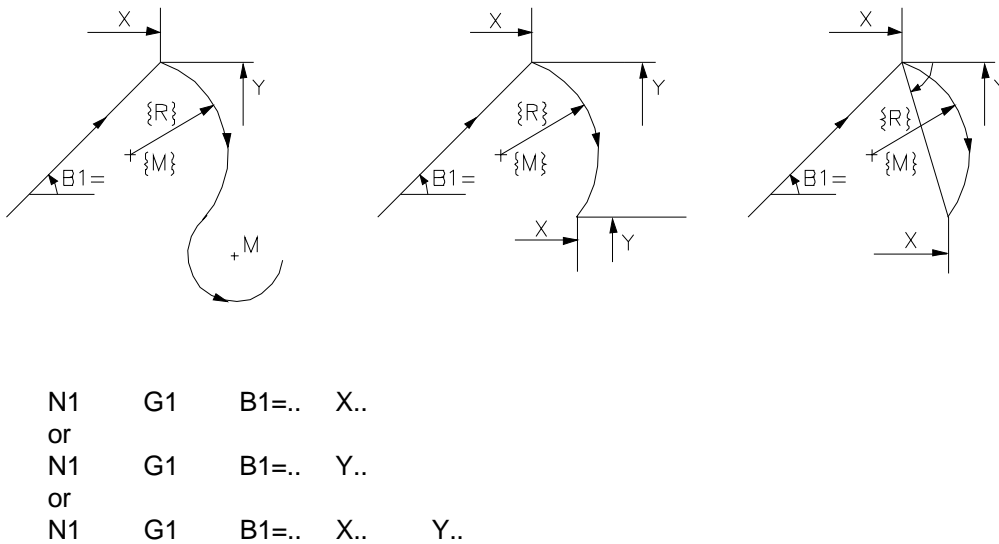
N1	G1	X.. or	Y.. or	X..	Y..
N2	G2/G3	R..		B1=..	X.. or Y..
or					
N1	G1	X.. or	Y.. or	X..	Y..
N2	G2/G3	I..	J..		B1=.. X.. or Y..

INTERSECTION POINT



Start point from N1 is not known

If the start point from N1 is not known, the angle, which the line makes with the main axis, has to be programmed in block N1 too. So this block reads:



Block N2 from the mentioned cases remains the same.

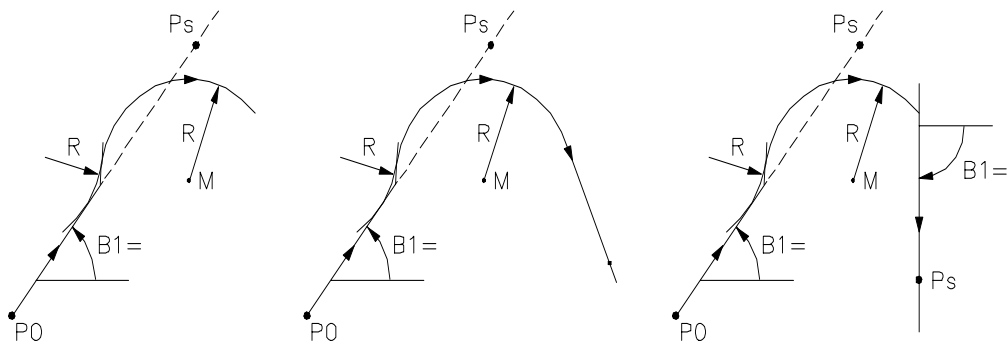
20.2.8 Rounding between intersecting line and circle

To insert a rounding between an intersecting line and a circle

Notice that the direction of rotation of the rounding is opposite to that of the programmed circle.

calculated intersection point

Start point from N1 is known

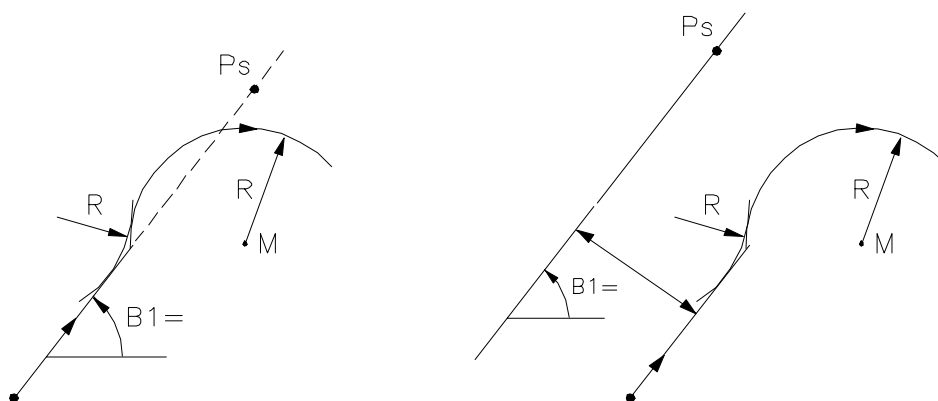


N1	G1	B1=..	J1=1/2
N2	G3/G2	R..	
N3	G2/G3	etc.	
or			
N1	G1	X..	Y.. I1=0 J1=1/2
N2	G3/G2	R..	
N3	G2/G3	etc.	

Refer to the section for calculating the intersection point for the formats of block N3.

Start point from N1 is not known

If the start point from N1 is not known, both the angle and a support point have to be programmed in block N1, So this block reads:

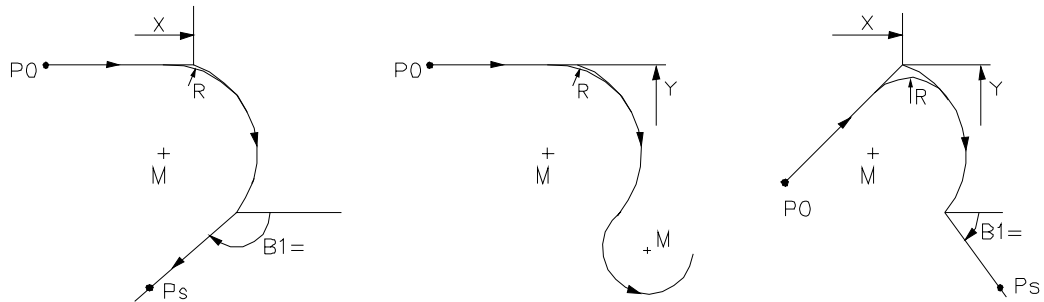


N1	G1	B1=..	X..	Y..	I1=0 or	I1=..	J1=1/2
N2	G3/G2	R..					
N3	G2/G3	etc.					

Refer to the section with the known start point for the formats of block N3.

Programmed intersecting point

Start point from N1 is known



N1	G1	X.. or Y.. or X..	Y..
N2	G3/G2	R..	
N3	G2/G3	etc.	

Refer to the section for programming the intersection point for the formats of block N3.

Start point from N1 is not known

If the start point from N1 is not known, the angle, which the line makes with the main axis, has to be programmed in block N1 too. So this block reads:

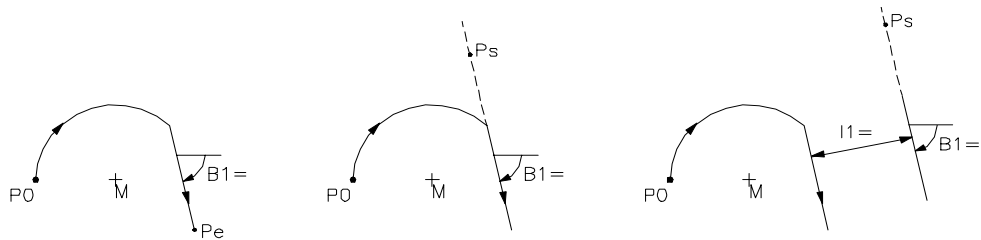
N1	G1	B1=..	X.. or Y.. or X..	Y..
N2	G3/G2	R..		
N3	G2/G3	etc.		

Refer to the section with the known start point for the formats of block N3..

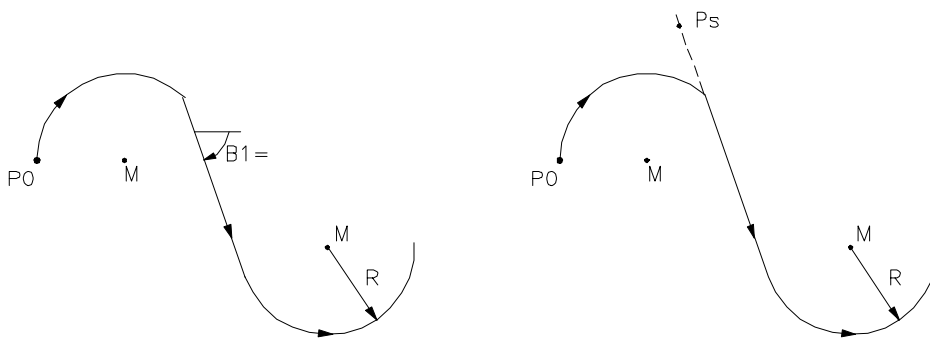
20.2.9 Intersecting point between circle and line

To calculate the intersection point between circle and line

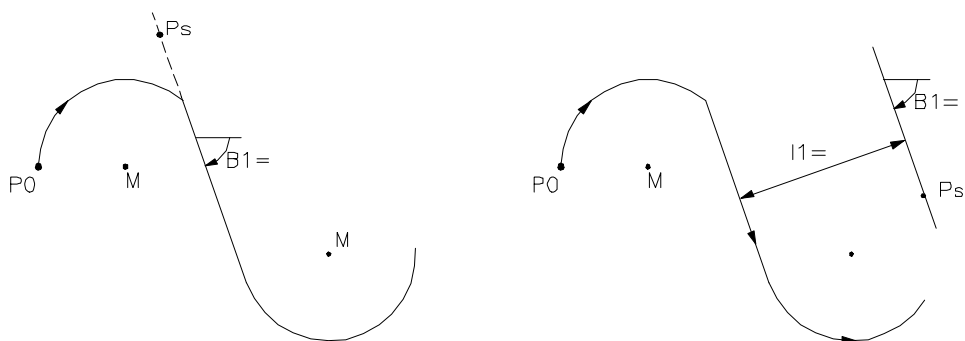
Start point from N1 is known



N1	G2/G3	I..	J..	J1=1/2
N2	G1	B1=.. X..	Y..	
or				
N1	G2/G3	I..	J..	J1=1/2
N2	G1	B1=.. X..	Y..	l1=0 or l1=..

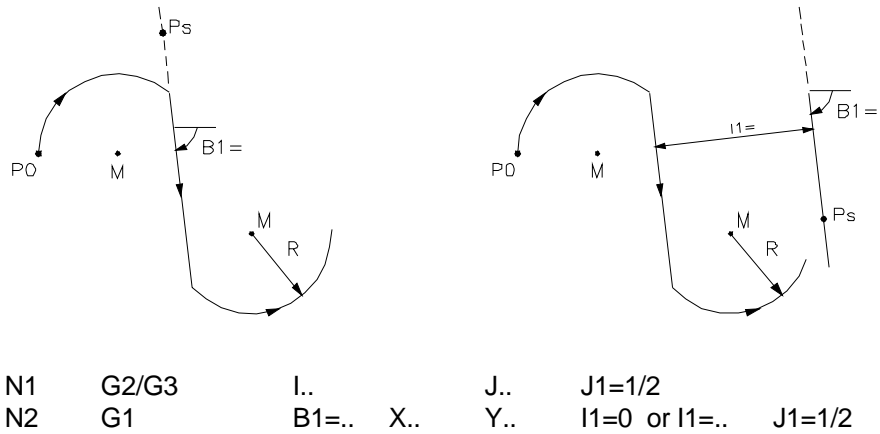


N1	G2/G3	I..	J..	J1=1/2
N2	G1	B1=.. R1=0		
or				
N1	G2/G3	I..	J..	J1=1/2
N2	G1	X..	Y..	l1=0 R1=0



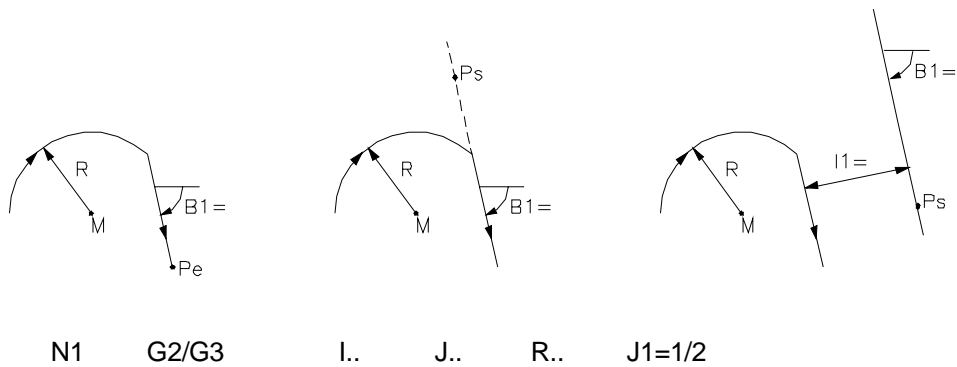
N1	G2/G3	I..	J..	J1=1/2
N2	G1	B1=.. X..	Y..	l1=0 or l1=.. R1=0

INTERSECTION POINT



Start point from N1 is not known

If the start point is not known, the radius value (R-word) should also be programmed in the block N1, thus this block reads:

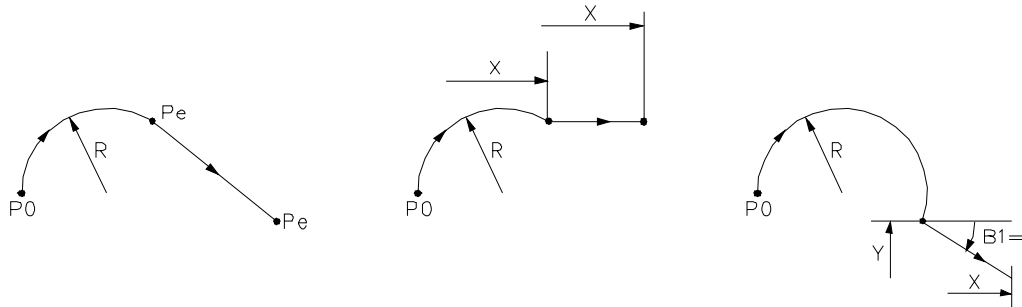


Block N2 from the mentioned cases remains the same.

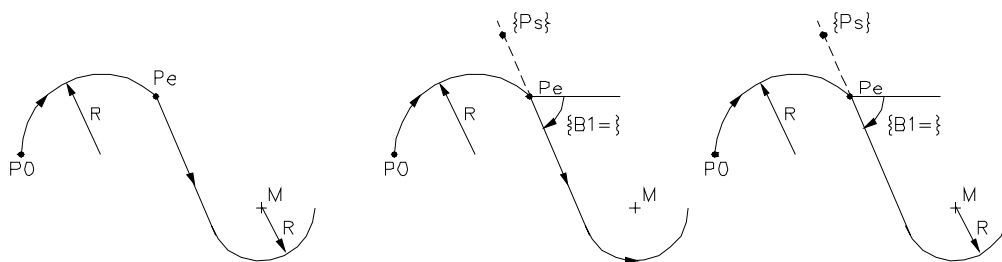
20.2.10 Intersecting point of circle and line programmed as end point

In some cases the intersection point of the circle and line is known from the drawing and can be programmed directly. It is assumed that this point is the start point of the next movement. The following formats are possible:

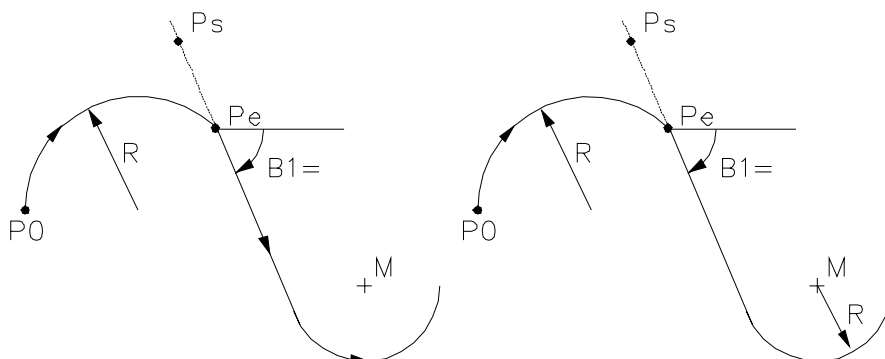
Start point from N1 is known



N1	G2/G3	R..	[end point]
N2	G1		[end point]



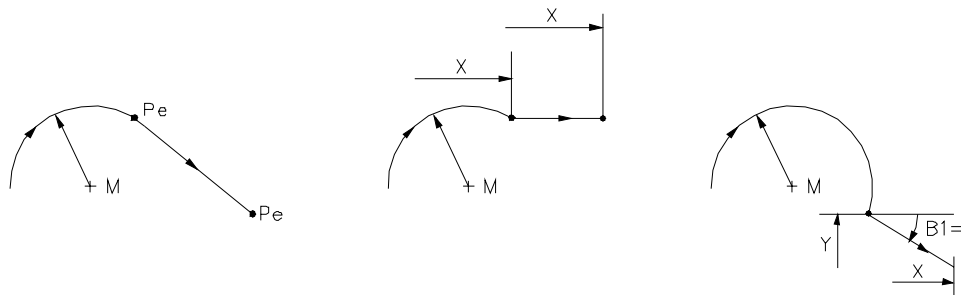
N1	G2/G3	R..	[end point]
N2	G1	R1=0	
or			
N1	G2/G3	R..	[end point]
N2	G1	B1=..	{R1=0} {J1=1/2}
or			
N1	G2/G3	R..	[end point]



N2	G1	X..	Y..	I1=0	{R1=0} {J1=1/2}
N1	G2/G3	R..		[end point]	
N2	G1	B1=..	X..	Y..	I1=0 or I1=.. {R1=0} {J1=1/2}

Start point from N1 is not known

If the start point from N1 is not known, the centre point coordinates instead of the radius have to be programmed in block N1. So this block reads:



N1 G2/G3 I.. J.. [end point]

Block N2 from the mentioned cases remains the same.

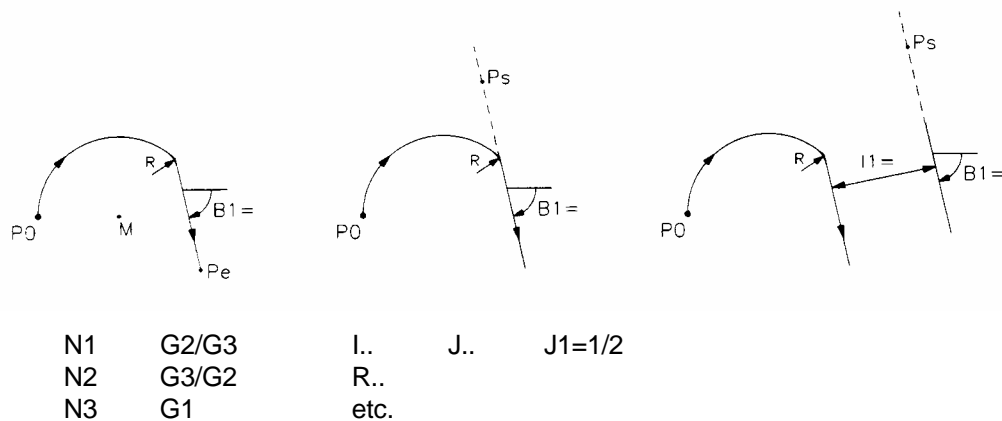
20.2.11 Rounding between intersecting circle and line

To insert a rounding between an intersecting circle and a line.

Notice that the direction of rotation of the rounding is opposite to that of the programmed circle.

Calculated intersection point

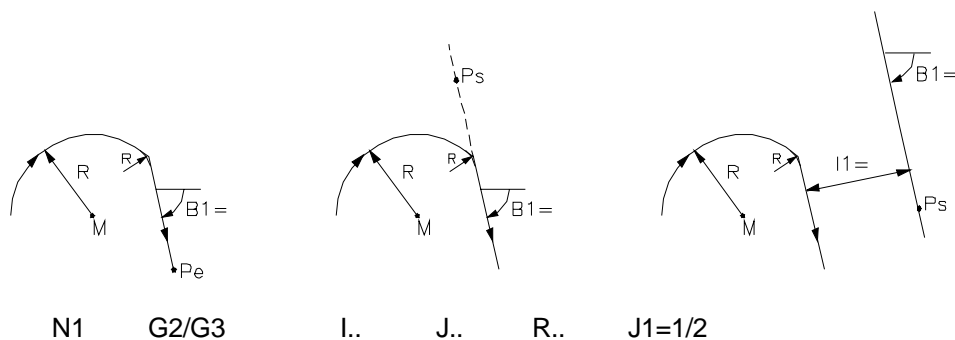
Start point from N1 is known



Refer to the section for calculating the intersection point for the formats of block N3

Start point from N1 is not known

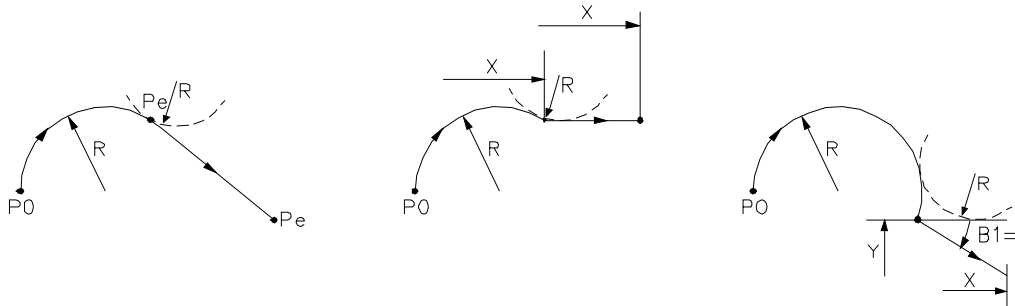
If the start point is not known, the value (R-word) should also be programmed in the block N1, thus this block reads:



Refer to the section with the known start point for the formats of block N3

Programmed intersecting point

Start point from N1 is known



N1	G2/G3	R..	[end point]
N2	G3/G2	R..	
N3	G1	etc.	

Refer to the section for programming the intersection point for the formats of block N3.

Note: A rounding can only be inserted if both the circle and the line are programmed with the endpoint, as indicated in the first format.

Start point from N1 is not known

If the start point from N1 is not known, the centre point coordinates instead of the radius have to be programmed in block N1. So this block reads:

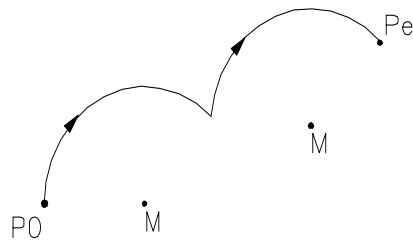
N1	G2/G3	I..	J..	[end point]
N2	G3/G2	R..		

Refer to the section with the known start point for the formats of block N3..

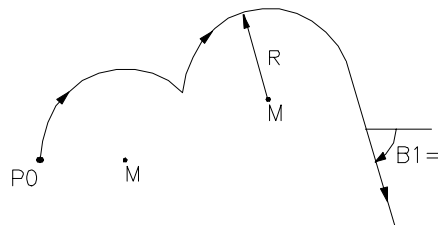
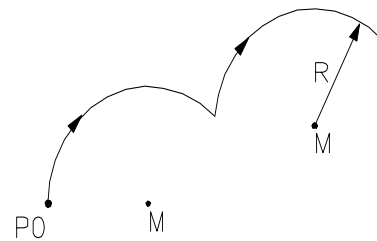
20.2.12 Intersecting point between two circles

To calculate the intersection point between two circles

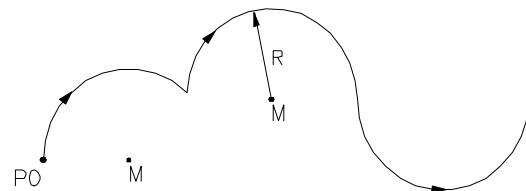
Start point from N1 is known



N1	G2/G3	I..	J..	J1=1/2	
N2	G2/G3	I..	J..	X..	Y..
or					
N1	G2/G3	I..	J..	J1=1/2	
N2	G2/G3	I..	J..	R..	



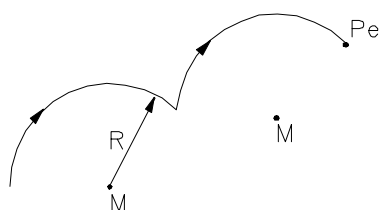
N1	G2/G3	I..	J..	J1=1/2	
N2	G2/G3	I..	J..	R..	R1=0



N1	G2/G3	I..	J..	J1=1/2	
N2	G2/G3	I..	J..	R..	J1=1/2

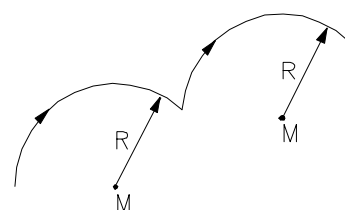
Start point from N1 is not known

If the start point is not known, the radius value (R-word) should also be programmed in the block N1, thus this block reads:



N1	G2/G3	I..	J..	R..	J1=1/2
----	-------	-----	-----	-----	--------

The N2 block remains unchanged.

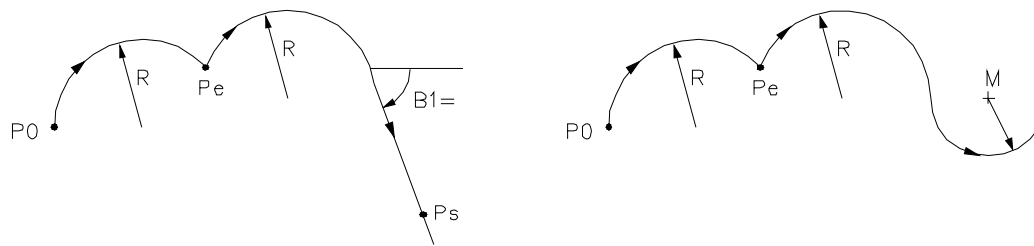


20.2.13 Intersection point between two circles programmed as end point

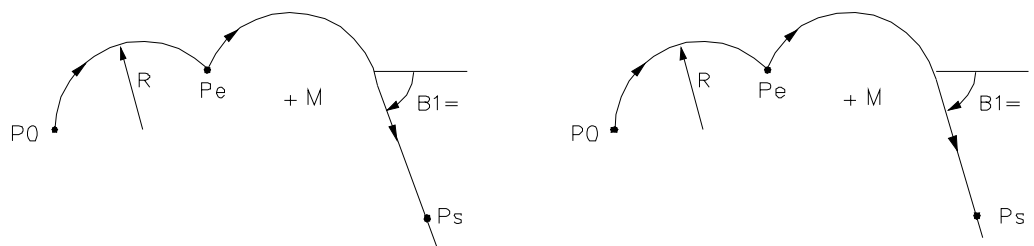
The intersection point between two circles is sometimes shown in the drawing and can be programmed directly. This point is assumed to be the start point of the next movement.

Start point from N1 is known

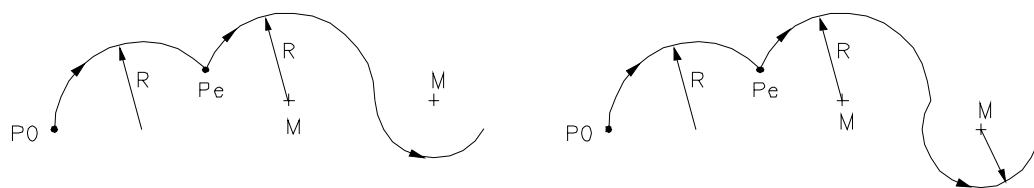
If the start point is not known, the radius value (R-word) should also be programmed in the block N1, thus this block reads:



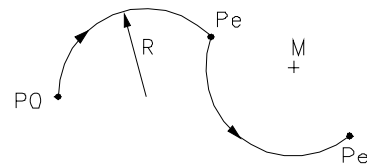
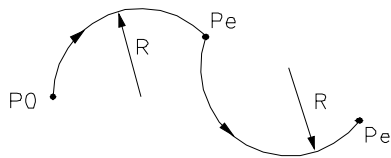
N1	G2/G3	R..	[end point]
N2	G2/G3	R..	R1=0



N1	G2/G3	R..	[end point]
N2	G2/G3	I..	J.. {R1=0} {J1=1/2}



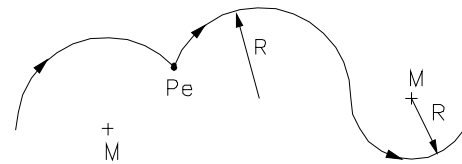
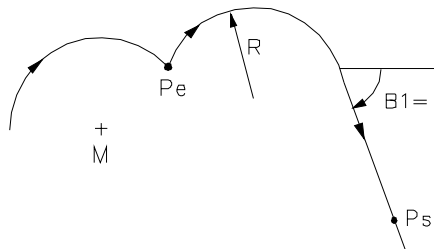
N1	G2/G3	R..	[end point]
N2	G2/G3	I..	J.. R.. {R1=0} {J1=1/2}



N1	G2/G3	R..	[end point]
N2	G2/G3	R..	[end point]
or			
N1	G2/G3	R..	[end point]
N2	G2/G3	I..	J.. [end point]

Start point from N1 is not known

If the start point from N1 is not known, the centre point coordinates instead of the radius have to be programmed in block N1. So this block reads:



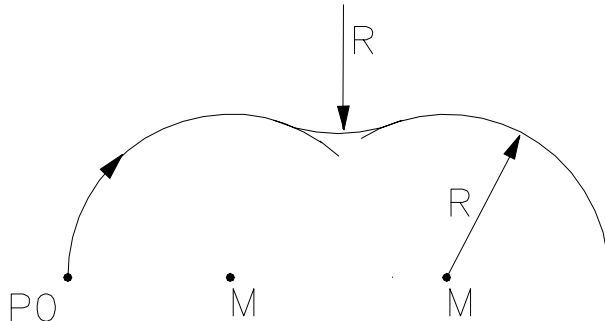
N1	G2/G3	I..	J.. [end point]
----	-------	-----	-----------------

Block N2 from the mentioned cases remains the same.

20.2.14 Rounding between two intersecting circles

Calculated intersection point

Start point from N1 is known



N1	G2/G3	I..	J..	J1=1/2
N2	G3/G2	R..		
N3	G2/G3	etc.		

Refer to the section for calculating the intersection point for the formats of block N3.

Start point from N1 is not known

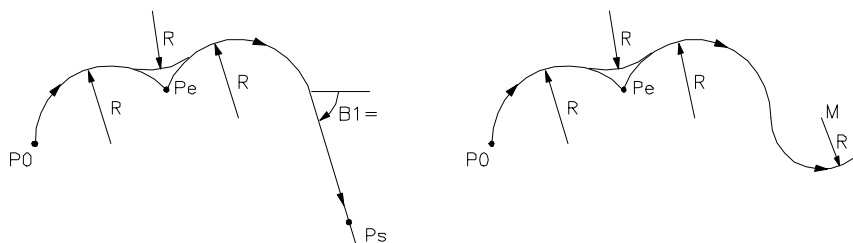
If the start point is not known, the radius value (R-word) should also be programmed in the block N1, thus this block reads:

N1	G2/G3	I..	J..	R..	J1=1/2
N2	G3/G2	R..			
N3	G2/G3	etc.			

Refer to the section with the known start point for the formats of block N3..

Programmed Intersection point

Start point from N1 is known



N1	G2/G3	R..	[End point]
N2	G3/G2	R..	
N3	G2/G3	etc.	

Refer to the section for programming the intersection point for the formats of block N3.

Start point from N1 is not known

If the start point from N1 is not known, the centre point coordinates instead of the radius have to be programmed in block N1. So this block reads:

N1	G2/G3	I..	J..	[end point]
N2	G3/G2	R..		
N3	G2/G3	etc.		

Refer to the section with the known start point for the formats of block N3.

20.3 Point of tangency

20.3.1 Point of tangency indicator (R1=)

A special word R1=0 is used to indicate that a geometric element is tangent to the next one (connecting circles are disregarded), thus:

- line tangent to circle
- circle tangent to line
- circle tangent to circle.

The word R1=0 is written in the block with the first element.

The point of tangency is chosen in such a way that the tool path is continue, that is to say the tool does not move backwards.

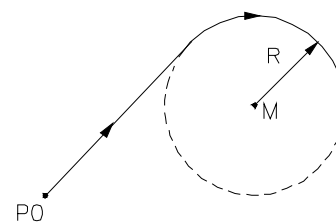
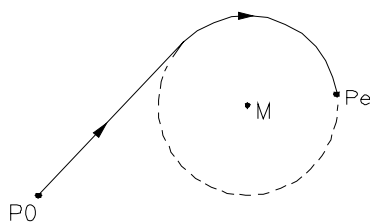
20.3.2 Tangent line and circle

To calculate the point of tangency between line and circle

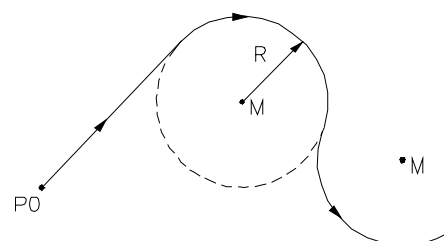
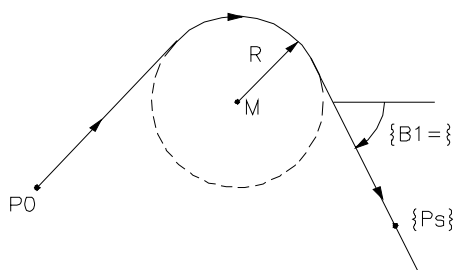
If the start point of the line is known, two cases must be considered:

only the start point from N1 is known

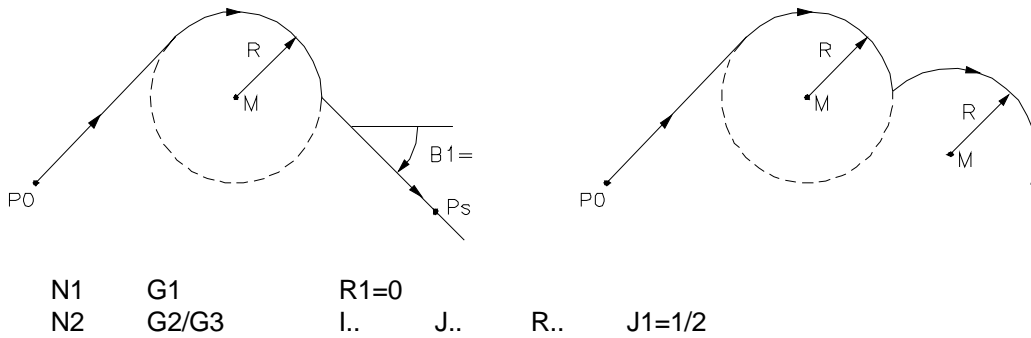
Its centre point coordinates and the radius or end point defines the circle. The following formats for line and circle can be used:



N1	G1	R1=0			
N2	G2/G3	I..	J..	X..	Y..
or					
N1	G1	R1=0			
N2	G2/G3	I..	J..	R..	

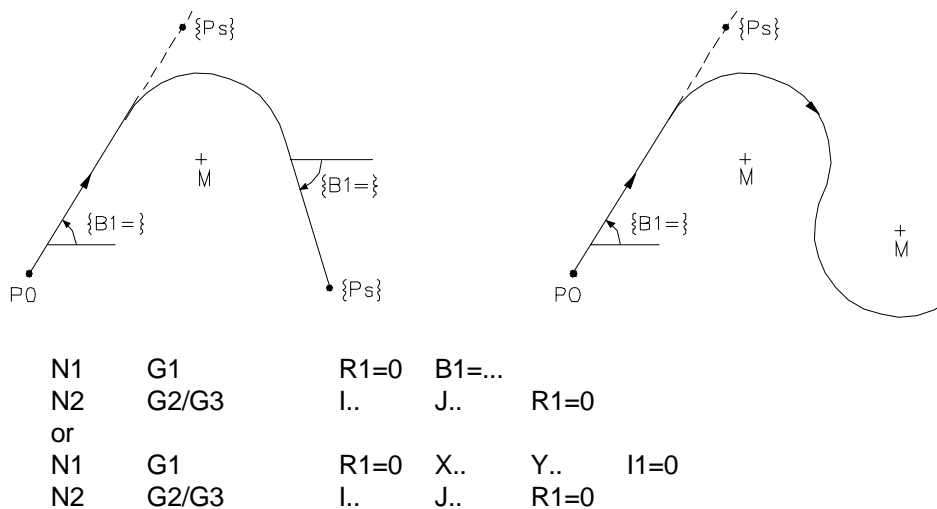
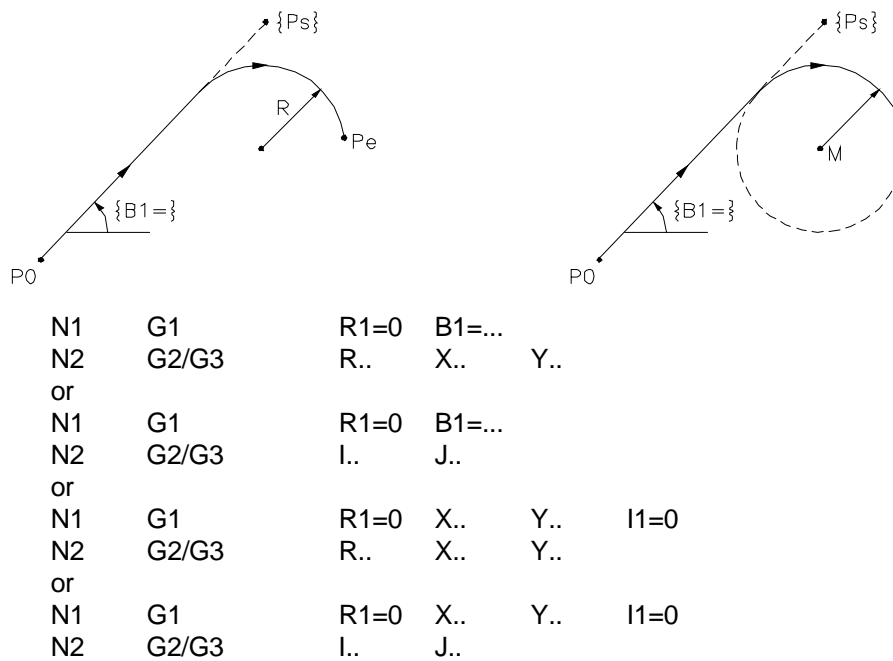


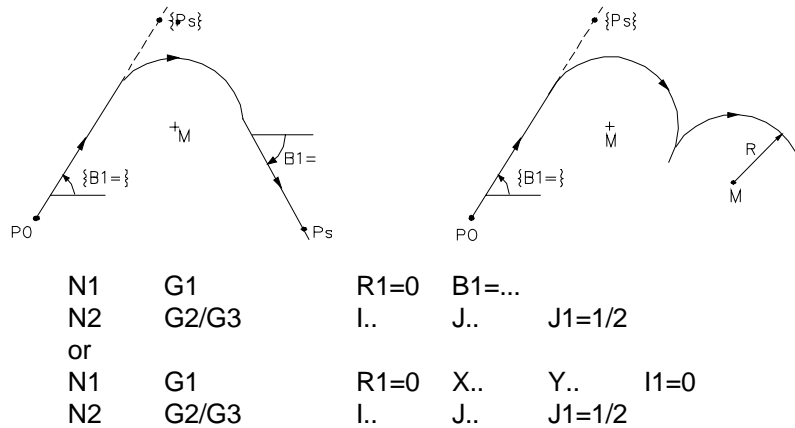
N1	G1	R1=0			
N2	G2/G3	I..	J..	R..	R1=0



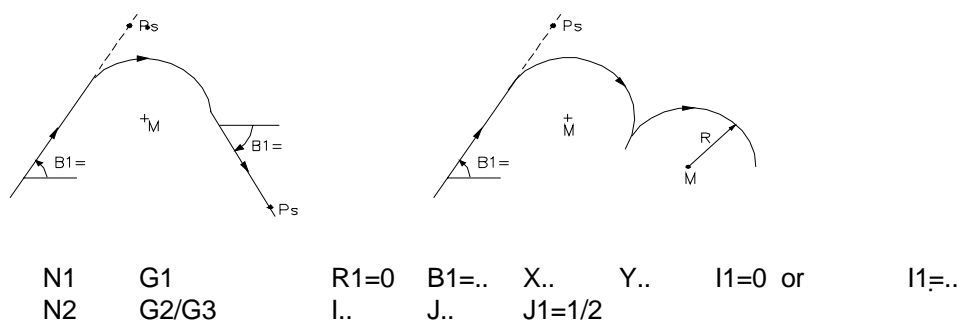
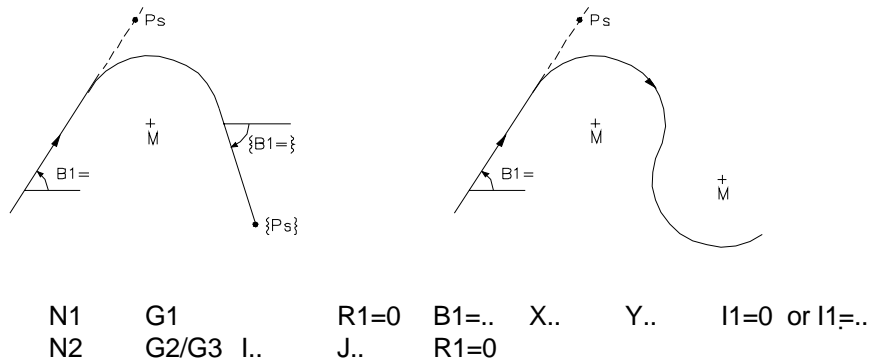
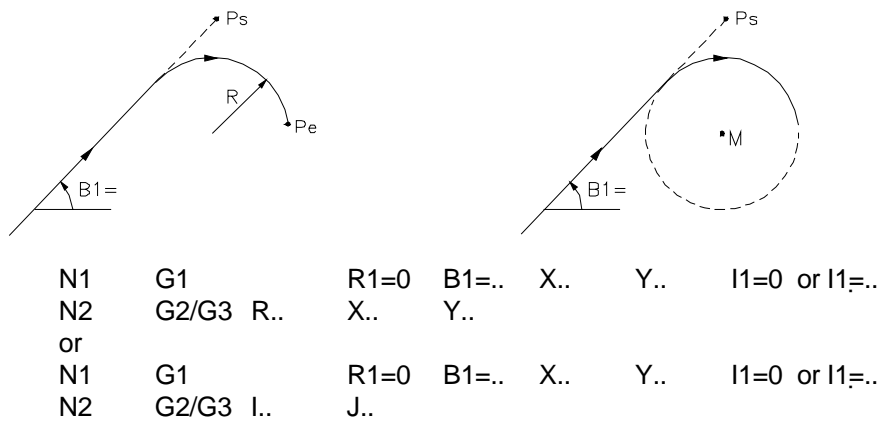
Start point from N1 and the angle with the axis or a support point on the line is known

From the circle in block N2 either the radius or the centre point coordinates must be calculated by the control. In this case the following formats are available:



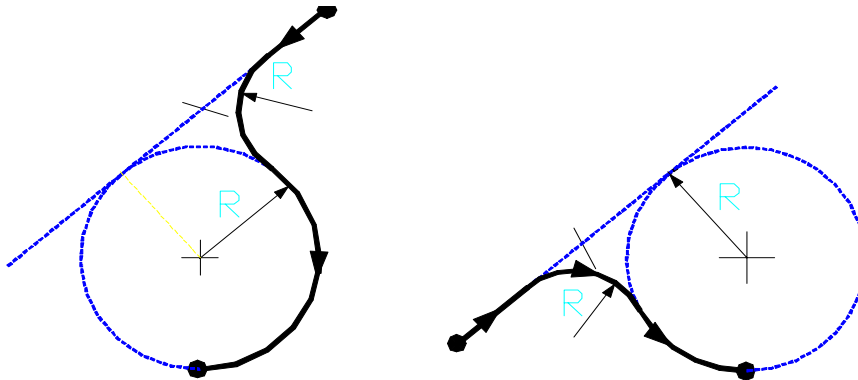
**Start point from N1 is not known**

If the start point of the line is not known, the angle with the main axis and a support point on the line have to be programmed. For block N2 the formats from the second case are used, thus:



20.3.3 Continuous connecting circle between tangent line and circle

To insert a connecting circle between a tangent line and a circle. Only one connecting circle is



possible. Its direction of rotation is opposite to the direction of rotation on the tangent circle.

Start point from N1 is known

N1	G1	R1=0	{B1=..}
N2	G2/G3	R..	
N3	G3/G2	etc.	
or			
N1	G1	R1=0	{X.. Y.. I1=0}
N2	G2/G3	R..	
N3	G3/G2	etc.	

Refer to the previous sections with a unknown start point from N1 for the formats of block N3

Start point from N1 is not known

N1	G1	R1=0	B1=..	X..	Y..	I1=0 or I1=..
N2	G2/G3	R..				
N3	G3/G2	etc.				

Refer to the section with unknown start point for the formats of block N3

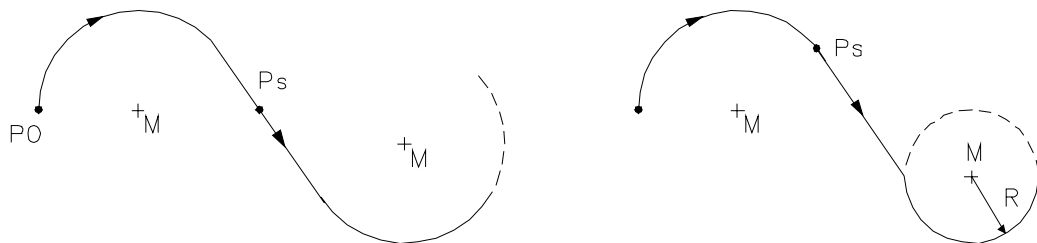
20.3.4 Tangent circle and line

To calculate the point of tangency between circle and line

Start point from N1 and the end point or a support point of the line is known

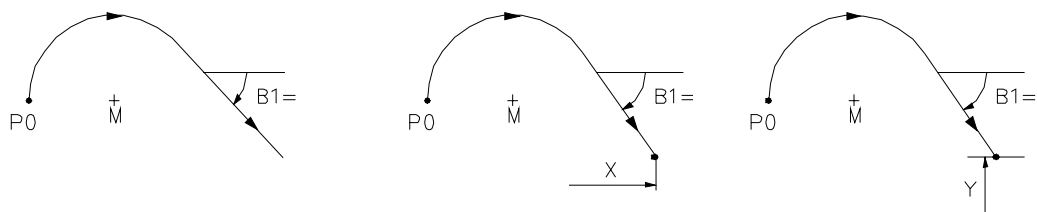


N1	G2/G3	I..	J..	R1=0	
N2	G1		X..	Y..	
or					
N1	G2/G3	I..	J..	R1=0	
N2	G1		X..	Y..	I1=0

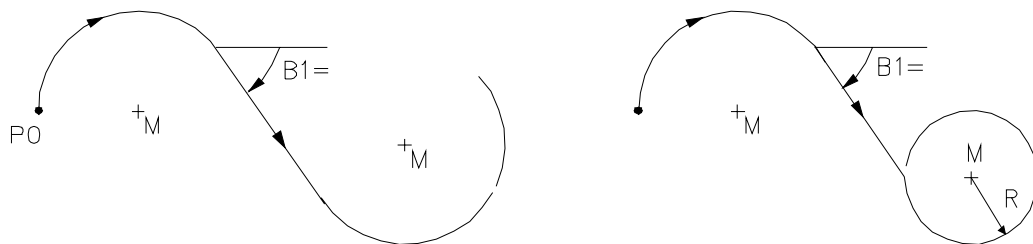


N1	G2/G3	I..	J..	R1=0	
N2	G1	X..	Y..	I1=0	R1=0
or					
N1	G2/G3	I..	J..	R1=0	
N2	G1	X..	Y..	I1=0	J1=1/2

Start point from N1 and the angle the line makes with the axis are known

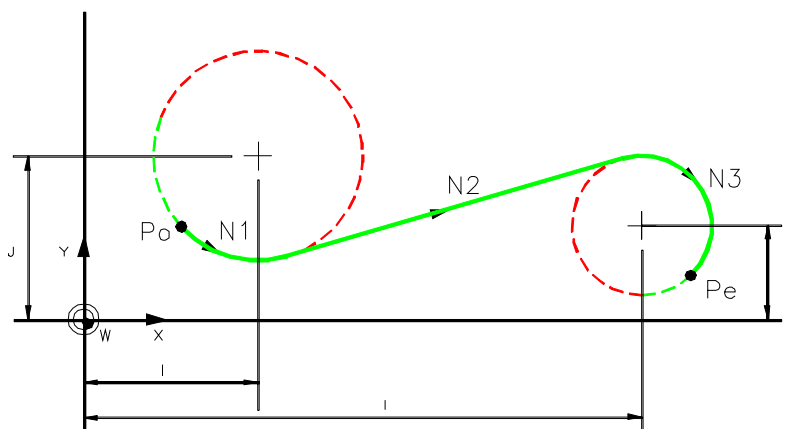


N1	G2/G3	I..	J..	R1=0	
N2	G1	B1=..			
or					
N1	G2/G3	I..	J..	R1=0	
N2	G1	B1=..	X..		
or					
N1	G2/G3	I..	J..	R1=0	
N2	G1	B1=..	Y..		



N1	G2/G3	I..	J..	R1=0
N2	G1	B1=..	R1=0	
or				
N1	G2/G3	I..	J..	R1=0
N2	G1	B1=..	J1=1/2	

Common tangent line of two circles



N1	G2/G3	I..	J..	R1=0
N2	G1	R1=0		
N3	G2/G3	I..	J..	R..
or				
N1	G2/G3	I..	J..	R1=0
N2	G1	R1=0		
N3	G2/G3	I..	J..	X.. Y..

Start point from N1 is not known

If the start point is not known, the radius value (R-word) should also be programmed in the block N1, thus this block reads:

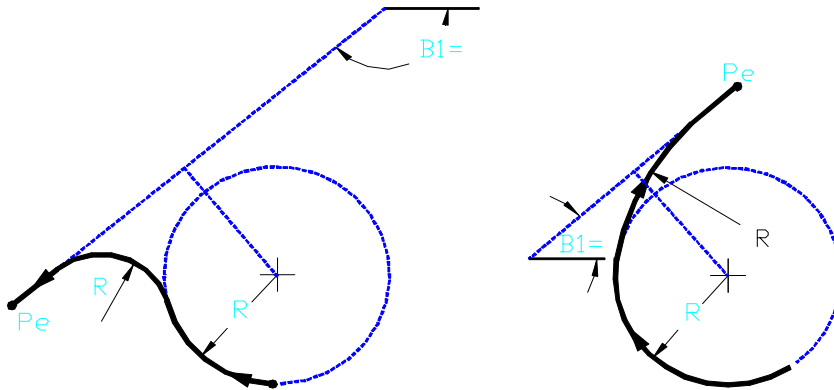


N1	G2/G3	I..	J..	R..	R1=0
----	-------	-----	-----	-----	------

The other blocks from the mentioned cases remain the same.

20.3.5 Continuous connection circle between tangent circle and line

To insert a connecting circle between a tangent circle and line. Only one connecting circle is possible. Its direction of rotation is opposite to the direction of rotation on the tangent circle



Start point from N1 is known

N1	G2/G3	I..	J..	R1=0
N2	G3/G2	R..		
N3	G1	etc.		

Refer to the previous sections with a known start point for the formats of block N3.

Start point from N1 is not known

If the start point is not known, the radius value (R-word) should also be programmed in the block N1, thus this block reads:

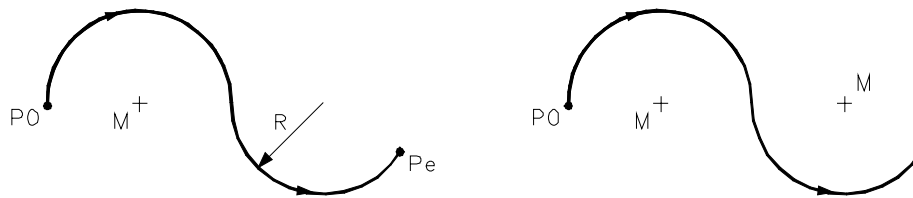
N1	G2/G3	I..	J..	R..	R1=0
N2	G3/G2	R..			
N3	G1	etc.			

Refer to the section with unknown start point for the formats of block N3.

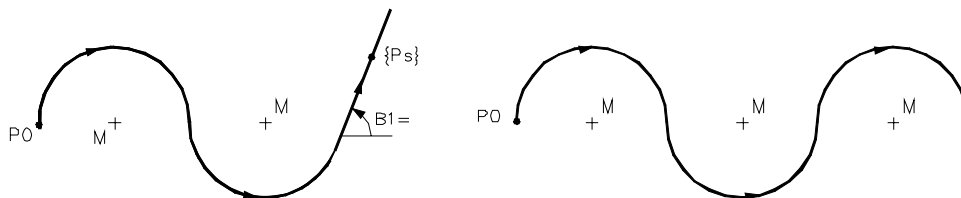
20.3.6 Tangent circle and line

To calculate the point of tangency between circle and line

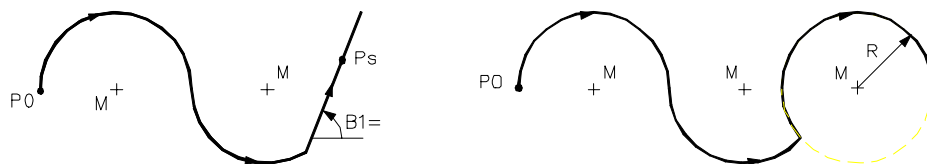
Start point from N1 is known



N1	G2/G3	I..	J..	R1=0
N2	G2/G3	R..	X..	Y..
or				
N1	G2/G3	I..	J..	R1=0
N2	G2/G3	I..	J..	



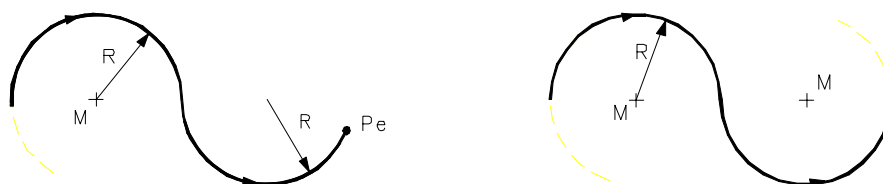
N1	G2/G3	I..	J..	R1=0
N2	G2/G3	I..	J..	R1=0



N1	G2/G3	I..	J..	R1=0
N2	G2/G3	I..	J..	J1=1/2

Start point from N1 is not known

If the start point is not known, the radius value (R-word) should also be programmed in the block N1, thus this block reads:



N1	G2/G3	I..	J..	R..	R1=0
----	-------	-----	-----	-----	------

The other blocks from the mentioned cases remain the same.

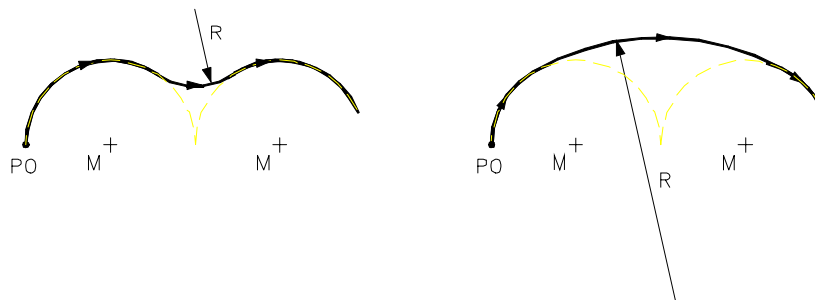
20.3.7 Continuous connecting circle between two tangent circles

To insert a connecting circle between two tangent circles. The connecting circle can be outside both circles or surround them. The programming of the direction of rotation on the three circles is:

1. Connecting circle tangent on the outside of both circles
G2, G3, G2 or G3, G2, G3
2. Connecting circle surrounds both circles
G2, G2, G2 or G3, G3, G3

For both cases the following formats are available. The right combination of the direction of rotation on the three circles has to be chosen by the programmer..

Start point from N1 is known



N1	G2/G3	I..	J..	R1=0
N2	G3/G2	R..		
N3	G2/G3	etc.		

Refer to the previous sections with a known start point for the formats of block N3.

Start point from N1 is not known

N1	G2/G3	I..	J..	R..	R1=0
----	-------	-----	-----	-----	------

The other blocks from the mentioned cases remain the same.

20.4 Continuous connecting circle between elements which do not meet

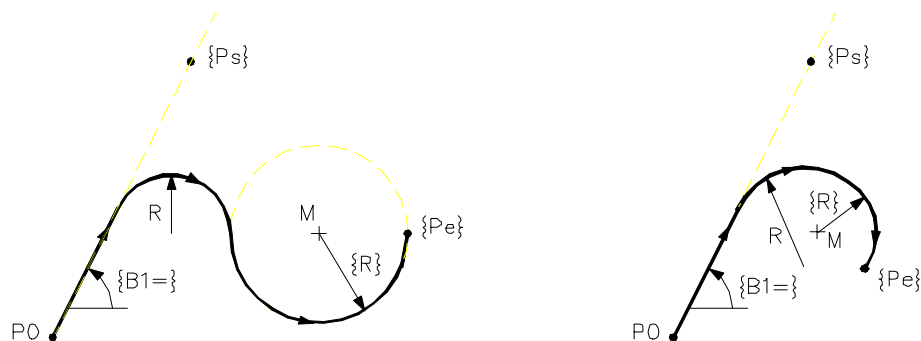
20.4.1 Line and circle

To insert a connecting circle between a line which does not meet a circle. Two connecting circles are possible:

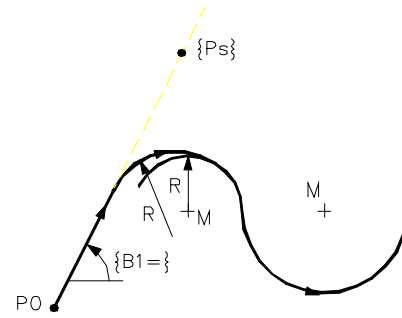
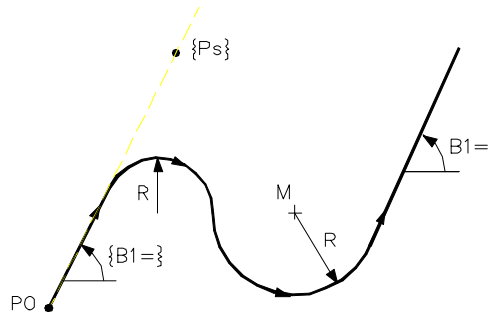
1. Connecting circle is tangent on the outside of the circle
G1, G2, G3 or G1, G3, G2
2. Connecting circle surrounds the circle
G1, G2, G2 or G1, G3, G3

For both cases the following formats are available. The right combination of the direction of rotation on the circles has to be chosen by the programmer.

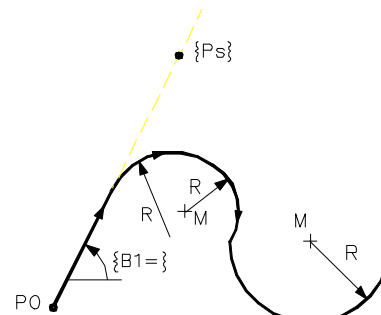
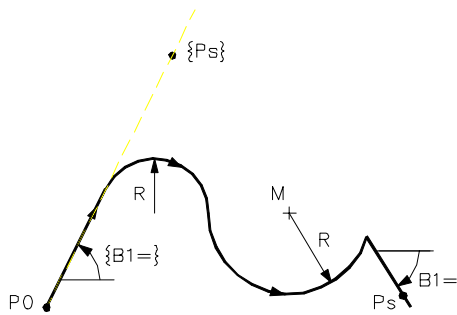
Start point from N1 is known



N1	G1	B1=..			
N2	G3/G2	R..			
N3	G2/G3	I..	J..	X..	Y..
or					
N1	G1	B1=..			
N2	G3/G2	R..			
N3	G2/G3	I..	J..	R..	
or					
N1	G1	X..	Y..	I1=0	
N2	G3/G2	R..			
N3	G2/G3	I..	J..	X..	Y..
or					
N1	G1	X..	Y..	I1=0	
N2	G3/G2	R..			
N3	G2/G3	I..	J..	R..	



N1	G1	B1=..			
N2	G3/G2	R..			
N3	G2/G3	I..	J..	R..	R1=0
or					
N1	G1	X..	Y..	I1=0	
N2	G3/G2	R..			
N3	G2/G3	I..	J..	R..	R1=0



N1	G1	B1=..			
N2	G3/G2	R..			
N3	G2/G3	I..	J..	R..	J1=1/2
or					
N1	G1	X..	Y..	I1=0	
N2	G3/G2	R..			
N3	G2/G3	I..	J..	R..	J1=1/2

Start point from N1 is not known

N1	G1	B1=...	X..	Y..	I1=0 or I1=..
----	----	--------	-----	-----	---------------

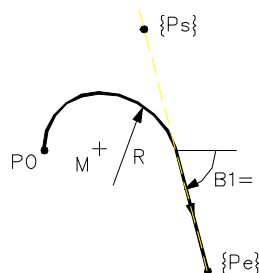
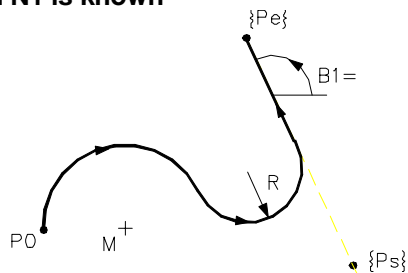
The other blocks from the mentioned cases remain the same.

20.4.2 Circle and line

To insert a connecting circle between a circle and a line which do not meet each other. Two connecting circles are possible:

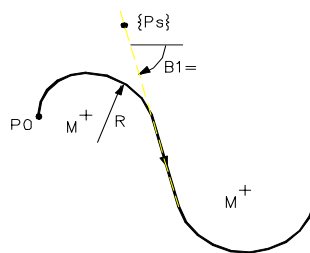
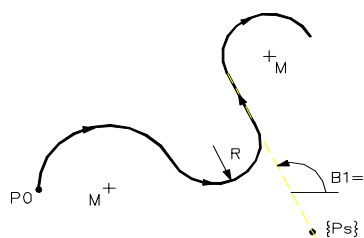
1. Connecting circle is tangent on the outside of the circle
G2, G3, G1 or G3, G2, G1
2. Connecting circle surrounds the circle
G2, G2, G1 or G3, G3, G1

Start point from N1 is known



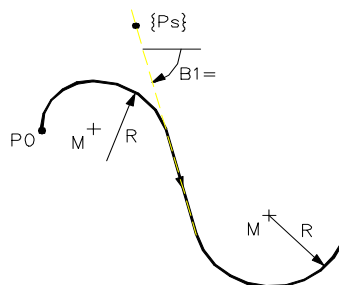
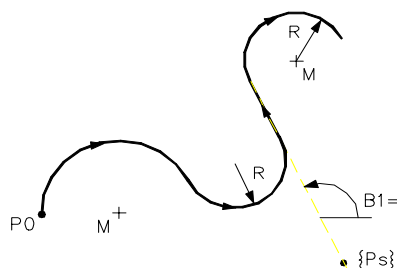
N1	G2/G3	I..	J..	
N2	G3/G2	R..		
N3	G1	B1=..	X..	Y..
or				
N1	G2/G3	I..	J..	
N2	G3/G2	R..		
N3	G1	B1=..	X..	Y..

l1=0



N1	G2/G3	I..	J..	
N2	G3/G2	R..		
N3	G1	B1=..	X..	Y..

l1=0 R1=0



N1	G2/G3	I..	J..	
N2	G3/G2	R..		
N3	G1	B1=..	X..	Y..

l1=0 J1=1/2

Start point from N1 is not known

N1	G2/G3	I..	J..	R..
----	-------	-----	-----	-----

The other blocks from the mentioned cases remain the same.

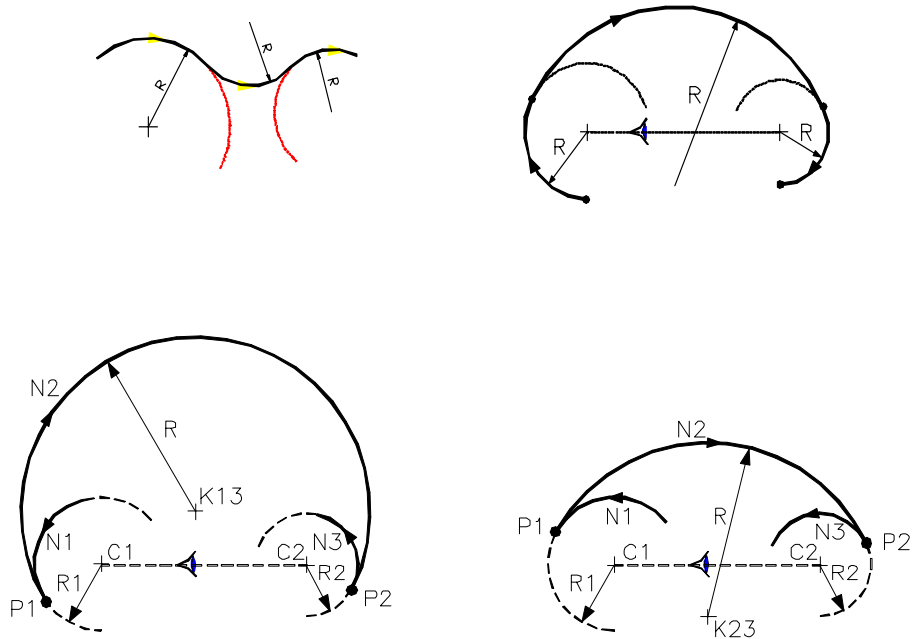
20.4.3 Two circles outside each other

To insert a connecting circle between two circles outside each other which do not meet. The direction of rotation on the three circles indicates the type of connecting circle, thus:

1. Connecting circle tangent on the outside of both circles
G2, G3, G2 or G3, G2, G3
2. Connecting circle surrounds both circles
G2, G2, G2 or G3, G3, G3
3. Connecting circle is tangent on the outside of the first circle and surrounds the second circle
G2, G3, G3 or G3, G2, G2
4. Connecting circle surrounds the first circle and is tangent on the outside of the second circle
G2, G2, G3 or G3, G3, G2

For all four cases the following formats are available. Any combination of the direction of rotation on the three circles is possible:

Start point from N1 is known



N1	G2/G3	I..	J..		
N2	G3/G2	R..			
N3	G2/G3	I..	J..	X..	Y..
or					
N1	G2/G3	I..	J..		
N2	G3/G2	R..			
N3	G2/G3	I..	J..	R..	{R1=0} {J1=1/2}

Start point from N1 is not known

N1	G2/G3	I..	J..	R..
----	-------	-----	-----	-----

The other blocks from the mentioned cases remain the same.

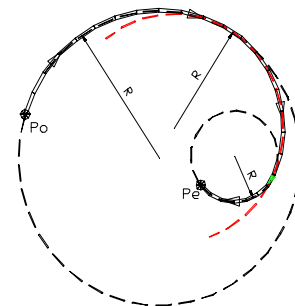
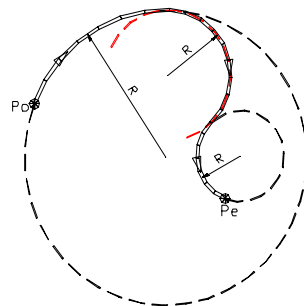
20.4.4 One circle inside the other one

To insert a connecting circle between a circle inside the other one which do not meet. The direction of rotation on the three circles indicates the type of connecting circle, thus:

1. Connecting circle tangent on the outside of inner circle
G2, G2, G3 or G3, G3, G2
2. Connecting circle tangent on the outside of inner circle
G2, G2, G2 or G3, G3, G3

For both cases the following formats are available:

Start point from N1 is known



N1	G2/G3	I..	J..		
N2	G2/G3	R..			
N3	G2/G3	I..	J..	X..	Y..
or					
N1	G2/G3	I..	J..		
N2	G2/G3	R..			
N3	G2/G3	I..	J..	R..	{R1=0} {J1=1/2}

Start point from N1 is not known

N1	G2/G3	I..	J..	R..
----	-------	-----	-----	-----

The other blocks from the mentioned cases remain the same.

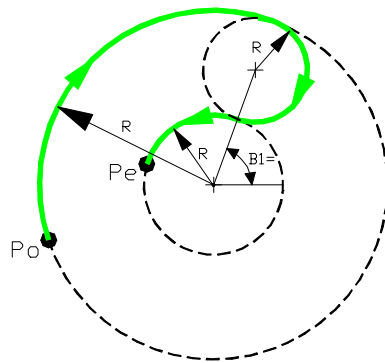
20.4.5 Concentric circles

Two concentric circles are a very special case of one circle inside the other one. In this case the centre points of both circles coincide, so some additional information has to be programmed to let the control calculate the connecting circle.

The word B1=.. which indicate the angle with the main axis of the line through the centre point of the concentric circles and the connecting circle, is used as additional information and has to be inserted in the block with the connecting circle.

For both cases the following formats are available:

Start point from N1 and radius of the connecting circle are known



N1	G2/G3	I..	J..	
N2	G2/G3	R..	B1=..	
N3	G2/G3	I..	J..	{R1=0} {J1=1/2}

Start point from N1 and radius of the second circle are known

In this case the radius of the connecting circle is calculated by the control.

N1	G2/G3	I..	J..		
N2	G2/G3	B1=..			
N3	G2/G3	I..	J..	X..	Y..

or

N1	G2/G3	I..	J..		
N2	G2/G3	B1=..			
N3	G2/G3	I..	J..	R..	{R1=0} {J1=1/2}

Start point from N1 is not known

N1	G2/G3	I..	J..	R..
----	-------	-----	-----	-----

The other blocks from the mentioned cases remain the same.

20.5 G64 Geometric calculations with non-continuous movements

Contents of this format section

- 4.1 Rounding or connecting circle indicator (K1=)
- 4.2 Rounding with intersection points
- 4.3 Rounding between intersecting straight lines
- 4.4 Rounding between intersecting line and circle
- 4.5 Rounding between intersecting circle and line
- 4.6 Rounding between two intersecting circles
- 4.7 Tangent lines (r1=)
- 4.8 Connecting circle between tangent line and circle or v.v.
- 4.9 Connecting circle between a line which do not meet a circle
- 4.10 Connecting circle between circles outside each other
- 4.11 Connecting circle between two circles one inside the other
- 4.12 Connecting circle with two concentric circles

20.5.1 Rounding or connecting circle indicator (K1=)

A special indicator (K1 =) is introduced to program which rounding or connecting circle is to be used.

For a rounding the value of K1= can be 1, 2, 3, or 4. Refer to the section ROUNDING WITH INTERSECTION POINTS for the meaning of these values. If with a rounding a wrong value is programmed, an error message is displayed.

For a connecting circle the indicator K1 = has two digits:

the first digit can have the value 1 or 2

- =1 the left connecting circle
- =2 the right connecting circle

the second digit indicates which connecting circle is meant and can have the values

- =0 with a line tangent to a circle or v.v.
- =0 or 1 with a line which does not meet a circle or v.v.
- =2 to 7 with circular elements

Refer to the proper sections for programming a connecting circle to see the meaning of left and right and of the second digit.

20.5.2 Rounding with intersection points

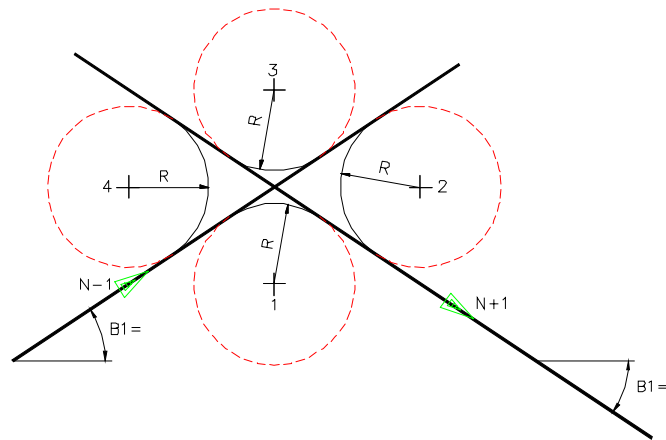
Between two intersecting elements a rounding can be inserted. In general four circles are possible which are numbered 1 to 4 and programmed with the word K1 = . The centre points of the circles with the numbers 1 and 2 are lying at the right from the first geometry element, when looking in the direction of the tool movement.

With the words K1 =2 or K1 =3 the contour intersects itself.

Note: If the word K1= is not programmed, a default value is used. This is K1=1 or K1=4 depending on the direction of movement on the second element.

20.5.3 Rounding between intersecting straight lines

To insert a rounding between two straight lines. The rounding can have any direction of rotation, thus G2 or G3.



Start point from N1 is known

N1	G1	B1=.
N2	G2/G3 R..	K1=1/2/3/4
N3	G1	etc.

Refer to the corresponding section with continuous movements for the formats of block N1 and N3.

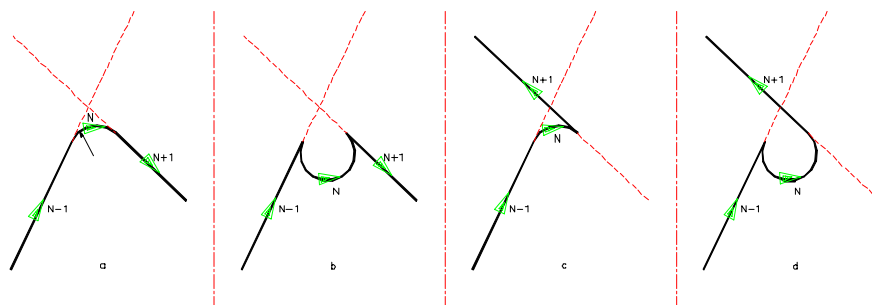
Start point from N1 is not known

If the start point from N1 is not known, both the angle and a support point have to be programmed in block N1. So this block reads:

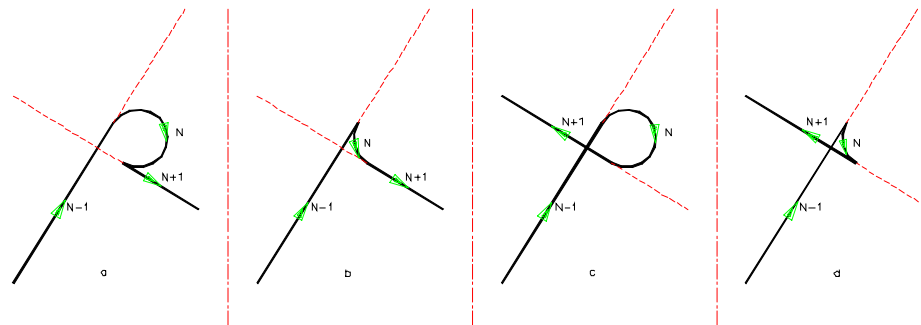
N1	G1	B1=.	X..	Y..	I1=0 or I1=.
N2	G2/G3 R..	K1=1/2/3/4			
N3	G1	etc.			

Refer to the corresponding section with continuous movements for the formats of block N1 and N3.

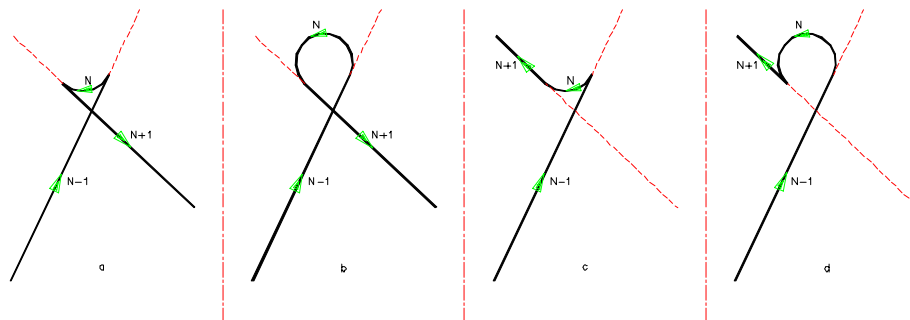
In the following illustrations the possibilities are shown with a circular connection between two straight lines.



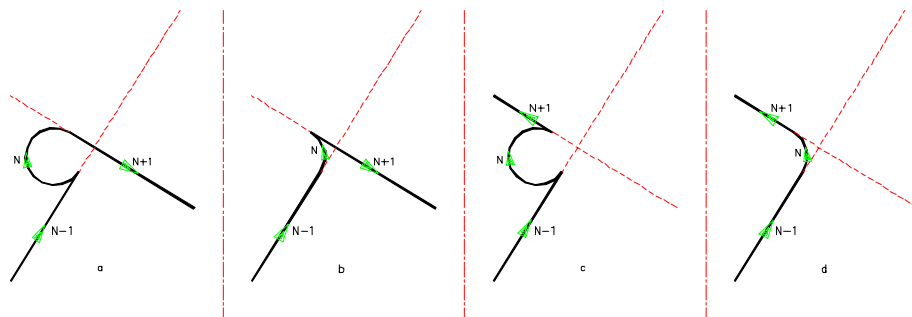
A circular connection with K1=1



A circular connection with $K1=2$



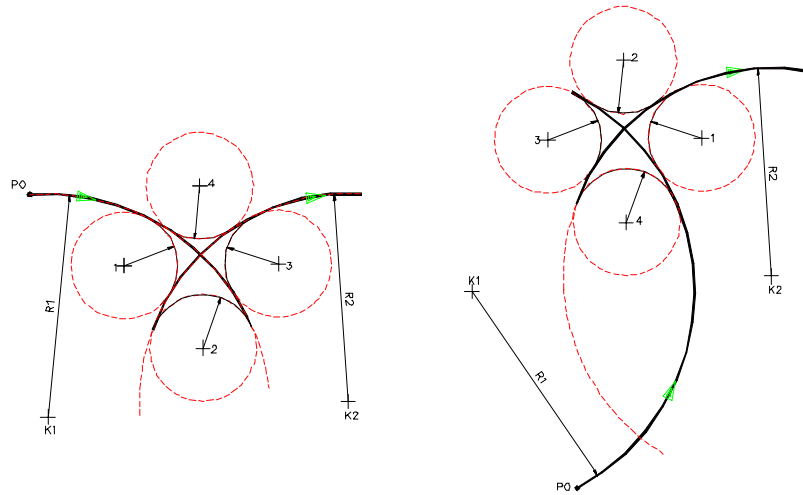
A circular connection with $K1=3$



A circular connection with $K1=4$

20.5.4 Rounding between intersecting line and circle

To insert a rounding between an intersecting line and a circle. The rounding can have any direction of rotation, thus G2 or G3.



Start point from N1 is known

N1	G1	B1=.	J1=1/2
N2	G3/G2	R..	K1=1/2/3/4
N3	G2/G3	etc.	

Start point from N1 is not known

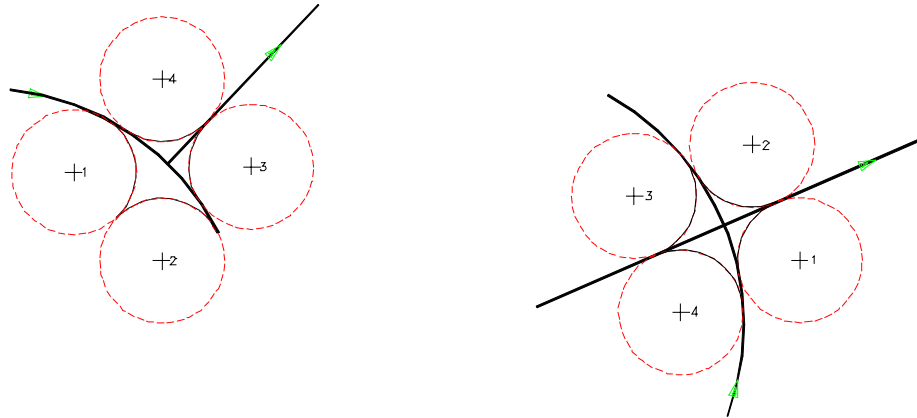
If the start point from N1 is not known, both the angle and a support point have to be programmed in block N1. So this block reads:

N1	G1	B1=.	X..	Y..	I1=0 or I1=.	J1=1/2
N2	G3/G2	R..	K1=1/2/3/4			
N3	G2/G3	etc.				

Refer to the corresponding section with continuous movements for the formats of block N1 and N3.

20.5.5 Rounding between intersecting circle and line

To insert a rounding between an intersecting circle and a line. The rounding can have any direction of rotation, thus G2 or G3.



Start point from N1 is known

N1	G2/G3	I..	J..	J1=1/2
N2	G3/G2	R..	K1=1/2/3/4	
N3	G1	etc.		

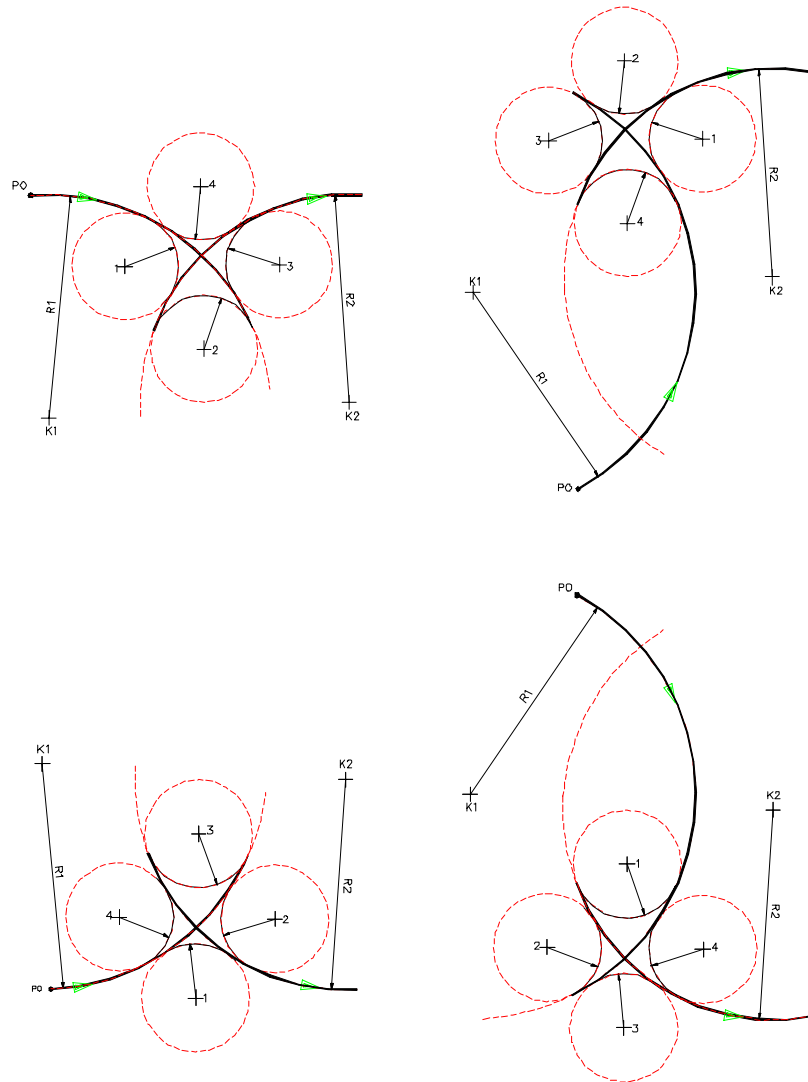
Start point from N1 is not known

N1	G2/G3	I..	J..	R..	J1=1/2
N2	G3/G2	R..	K1=1/2/3/4		
N3	G1	etc.			

Refer to the corresponding section with continuous movements for the formats of block N1 and N3.

20.5.6 Rounding between two intersecting circles

To insert a rounding between two intersecting circles. The rounding can have any direction of rotation, thus G2 or G3.



Start point from N1 is known

N1	G2/G3	I..	J..	J1=1/2
N2	G3/G2	R..	K1=1/2/3/4	
N3	G2/G3	etc.		

Start point from N1 is not known

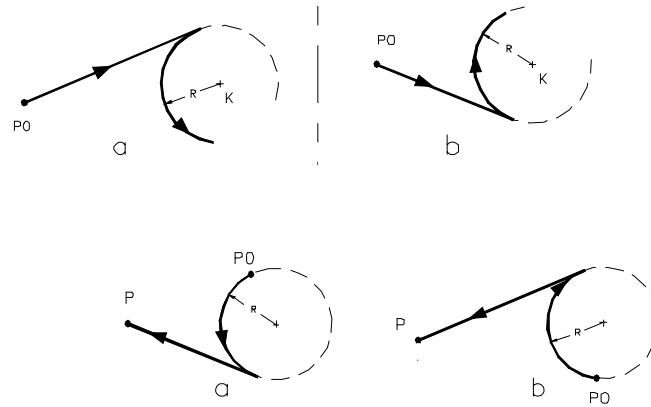
N1	G2/G3	I..	J..	R..	J1=1/2
N2	G3/G2	R..	K1=1/2/3/4		
N3	G2/G3	etc.			

Refer to the corresponding section with continuous movements for the formats of block N1 and N3.

20.5.7 Tangent lines (R1=)

From a point two lines can be drawn tangent to a circle. With the word R1= in the block with the tangent element is indicated which tangent line should be used:

- a. R1=1: the left tangent line
- b. R1=2: the right tangent line



Left and right are determined with a movement from:

line to circle by looking from start point to centre point

circle to line by looking from centre point to end point

The word R1=1 or R1=2 is programmed in the same way as explained for R1=0 in the continuous section.

Note: With R1=0 the control determines automatically which tangent line keeps the movement continuous, so the tool does not move backwards.

20.5.8 Connecting circle between a line tangent to a circle or v.v.

If a line is tangent to a circle two connecting circles are possible, one left of the line through the centre of the circle and perpendicular to the line and one to the right of that line.

The left circle is programmed with the indicator K1=10 and the right circle with K1=20.

Gerade tangiert an Kreis



Start point from N1 is known

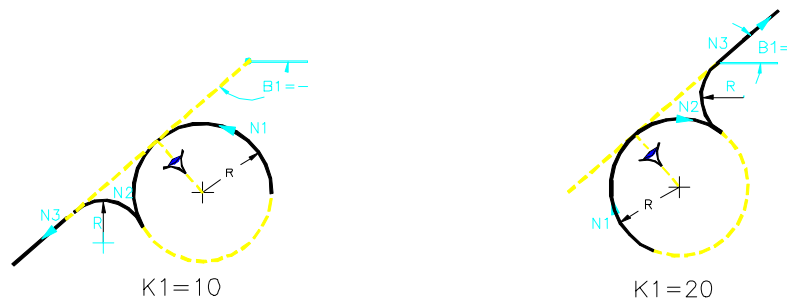
N1	G1		R1=1/2
N2	G2/G3	R..	K1=10/20
N3	G2/G3	etc.	

Start point from N1 is not known

N1	G1		B1=..	R1=1/2
N2	G2/G3	R..	K1=10/20	
N3	G2/G3	etc.		

Refer to the corresponding section with continuous movements for the formats of block N1 and N3.

Circle tangent to line



Start point from N1 is known

N1	G2/G3	I..	J..	R1=1/2
N2	G2/G3	R..	K1=10/20	
N3	G1	etc.		

Start point from N1 is not known

N1	G2/G3	I..	J..	R..	R1=1/2
N2	G2/G3	R..	K1=10/20		
N3	G1	etc.			

Refer to the corresponding section with continuous movements for the formats of block N1 and N3.

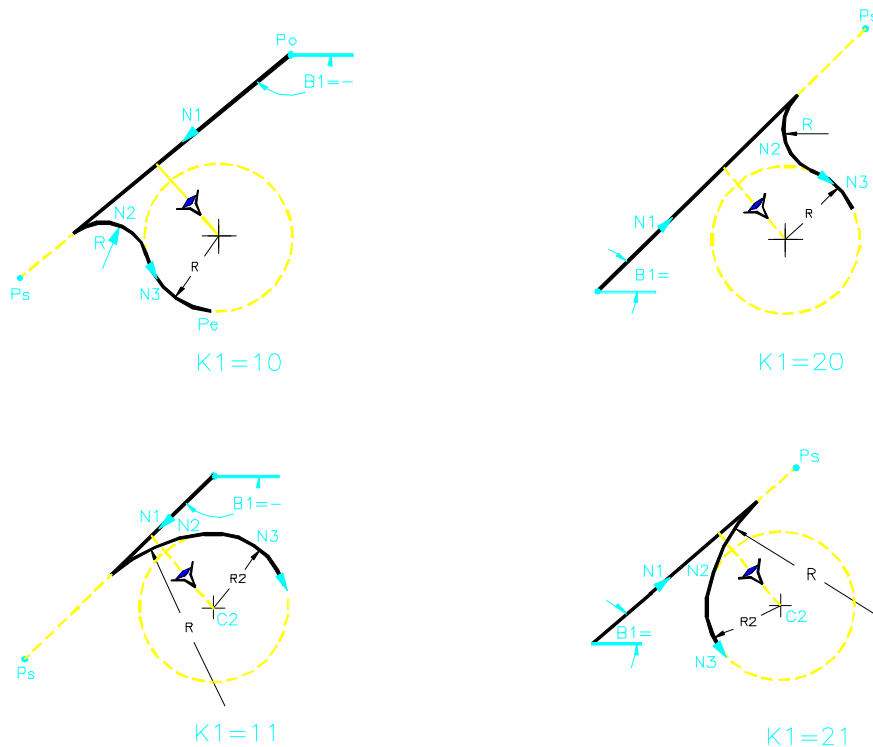
20.5.9 Connecting circle between a line which does not meet a circle

If a line does not meet a circle, two connecting circles are possible on the left of the line through the centre and perpendicular to the line. The same two circles are also possible to the right of that line. One circle touches the circle on the outside. The left circle is programmed with the word K1 =10 and the same circle on the right with K1 =20.

The second connecting circle surrounds the circle. In this case the left circle is programmed with the word K1=11 and the same circle on the right with K1 =21.

The formats are:

Line and circle



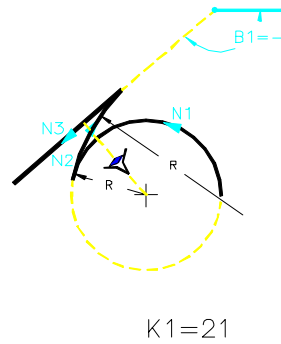
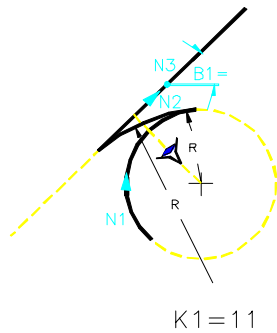
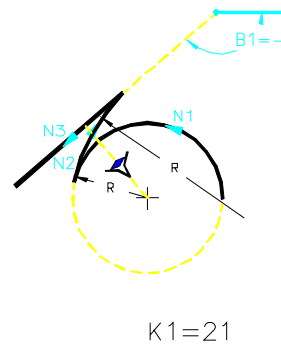
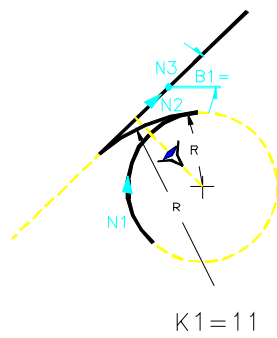
Start point from N1 is known

N1	G1	B1=..	{X.. Y.. I1=0}
N2	G2/G3	R..	K1=10/11 or K1=20/21
N3	G2/G3	etc.	

Start point from N1 is not known

N1	G1	B1=..	X.. Y.. I1=0
N2	G2/G3	R..	K1=10/11 or K1=20/21
N3	G2/G3	etc.	

Refer to the corresponding section with continuous movements for the formats of block N1 and N3.

Circle and line**Start point from N1 is known**

N1	G2/G3	I..	J..	R1=1/2
N2	G2/G3	R..	K1=10/11 or K1=20/21	
N3	G1	etc.		

Start point from N1 is not known

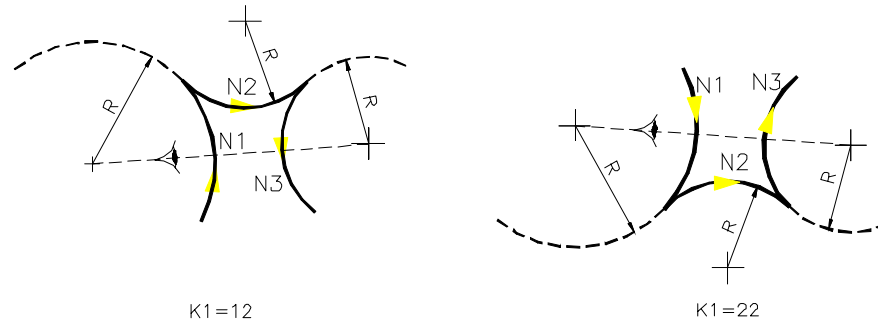
N1	G2/G3	I..	J..	R..	R1=1/2
N2	G2/G3	R..	K1=10/11 or K1=20/21		
N3	G1	etc.			

Refer to the corresponding section with continuous movements for the formats of block N1 and N3.

20.5.10 Connecting circle between circles outside each other

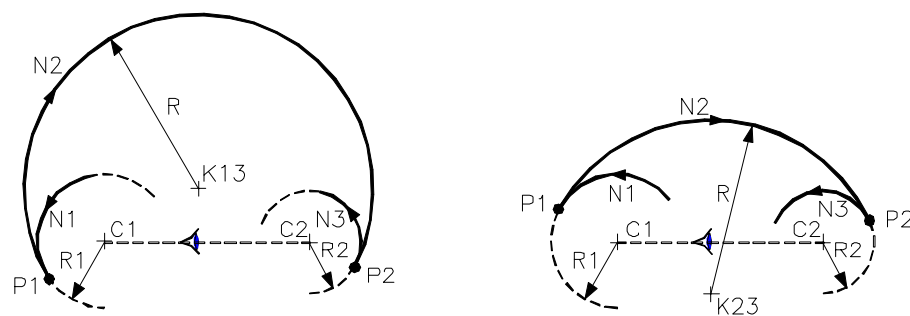
Four types of connecting circles on the left of the line through the centre points are possible with two circles, which do not meet and are outside each other. The first two types are also possible with tangent circles. The same four types can be found on the right of the line through the centres.

The word $K1=..$ for a connecting circle outside both circles is:



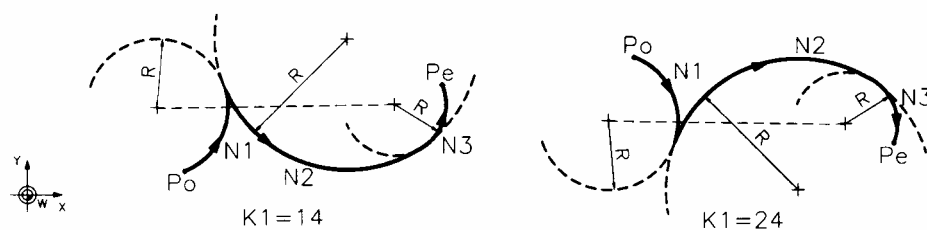
- for the **left** connecting circle: $K1 = 12$
- for the **right** connecting circle: $K1 = 22$

The word $K1=..$ for a connecting circle **surrounding both circles** is:



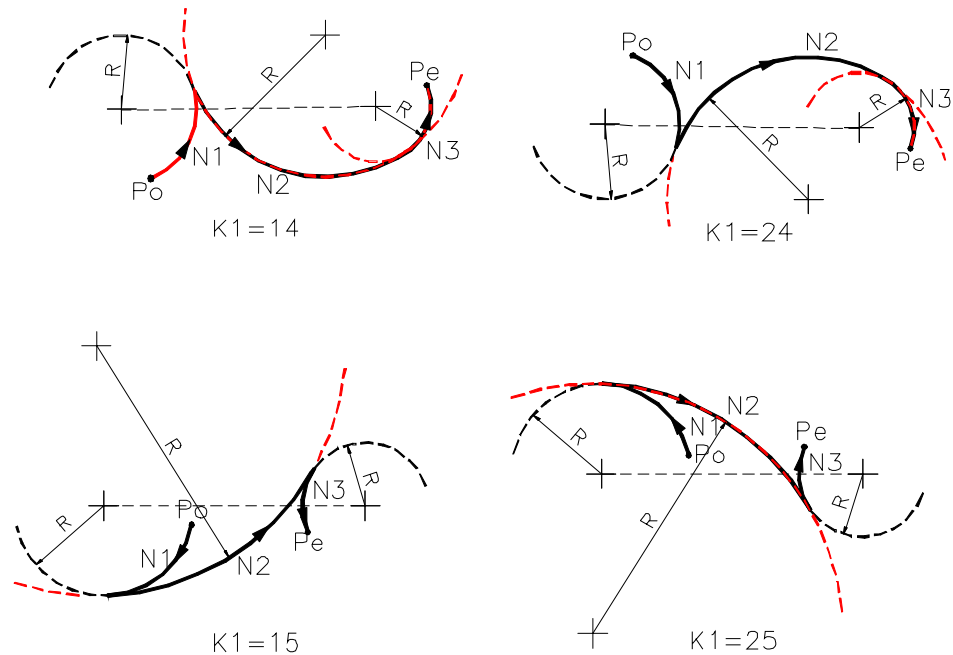
- for the **left** connecting circle: $K1=13$
- for the **right** connecting circle: $K1=23$

The word $K1 = ..$ for a connecting circle **outside the first circle** is:



- for the **left** connecting circle: $K1 = 14$
- for the **right** connecting circle: $K1=24$

The word K1 =.. for a connecting circle **surrounding the first circle** is:



- for the **left** connecting circle: K1=15
- for the **right** connecting circle: K1=25

Note: It depends on the programmed direction of movement (G2 and G3) on the three circles which default value for K1 = is used by the control.

The formats are:

Start point from N1 is known

N1	G2/G3	I..	J..	R1=1/2
N2	G2/G3	R..	K1=12/13/14/15 or K1=22/23/24/25	
N3	G2/G3	etc.		

Start point from N1 is not known

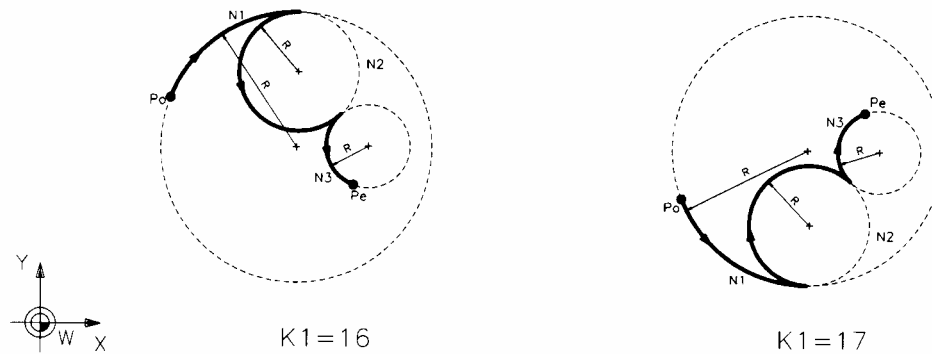
N1	G2/G3	I..	J..	R..	R1=1/2
N2	G2/G3	R..	K1=12/13/14/15 or K1=22/23/24/25		
N3	G2/G3	etc.			

Refer to the corresponding section with continuous movements for the formats of block N1 and N3.

20.5.11 Connecting circle between two circles one inside the other

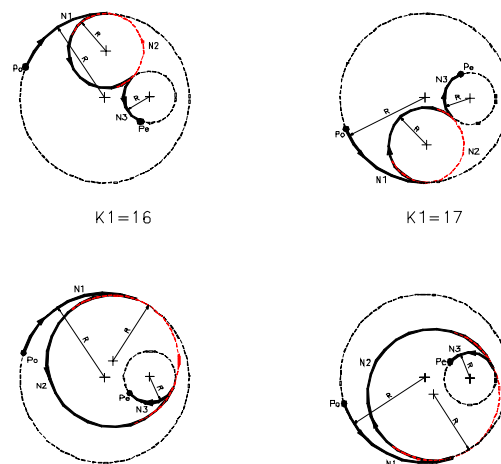
Two types of connecting circles on the left of the line through the centre points are possible with two circles of which one lies inside the other one. The same two types can be found on the right of the line through the centres.

The word K1 =.. for a connecting circle **outside the smaller circle** is:



- for the **left** connecting circle: K1=16
- for the **right** connecting circle: K1=26

The word K1=.. for a connecting circle **surrounding the smaller circle** is:



- for the **left** connecting circle: K1 =17
- for the **right** connecting circle: K1=27

Note: It depends on the programmed direction of movement (G2 and G3) on the three circles which default value for K1 = is used by the control.

The formats are:

Start point from N1 is known

N1	G2/G3	I..	J..	R1=1/2
N2	G2/G3	R..	K1=16/17 or K1=26/27	
N3	G2/G3	etc.		

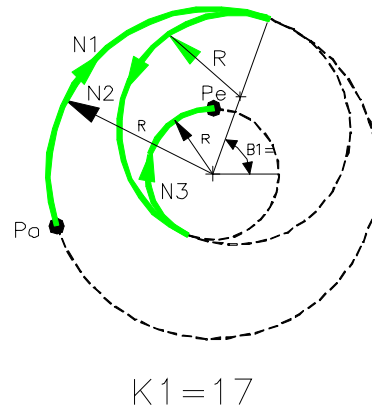
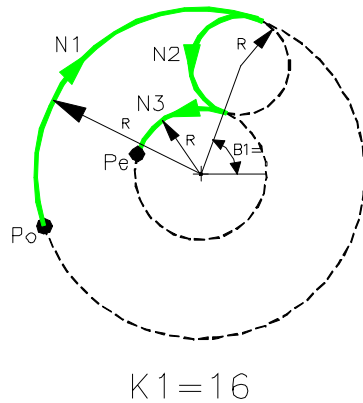
Start point from N1 is not known

N1	G2/G3	I..	J..	R..	R1=1/2
N2	G2/G3	R..	K1=16/17 or K1=26/27		
N3	G2/G3	etc.			

Refer to the corresponding section with continuous movements for the formats of block N1 and N3.

20.5.12 Connecting circle with two concentric circles

If both circles are concentric, the programming is the same as with one circle inside the other one, except that it is also necessary to program the angle (B1=..), which the line through the common centre point and the centre point of the connecting circle makes with the main axis.



For both cases the following format is available:

Start point from N1 and radius of the connecting circle are known

N1	G2/G3	I..	J..	R1=1/2
N2	G2/G3	R..	B1=..	K1=16/17
N3	G2/G3	I..	J..	

Start point from N1 and radius of the second circle are known

In this case the radius of the connecting circle is calculated by the control.

N1	G2/G3	I..	J..		
N2	G2/G3	B1=..	K1=16/17		
N3	G2/G3	I..	J..	X..	Y..

Refer to the corresponding section with continuous movements for the formats of block N1 and N3

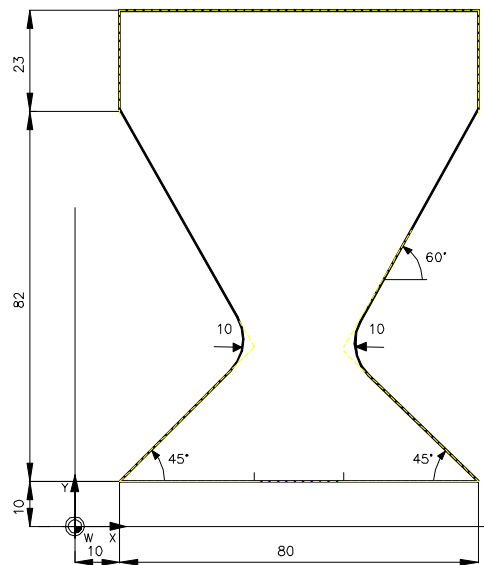
Start point from N1 is known

N1	G2/G3	I..	J..	R..
----	-------	-----	-----	-----

The other blocks from the mentioned cases remain the same.

20.6 Examples

Example 1 Calculation of intersection point



```

N64001
N1 G54
N2 S1000 T1 M6 (RADIUS 2 mm)
N3 G0 X0 Y0 Z10 M3
N4 G1 Z-10 F500
N5 G43 Y10
N6 G42
N7 G64
N8 X90
N9 B1=135

```

```

N10 G2 R10
N11 G1 X90 Y92 B1=60
N12 Y115
N13 X10
N14 Y92
N15 B1=-60

```

```

N16 G2 R10
N17 G1 X10 Y10 B1=45
N18 G40
N19 G63
N20 X0 Y0
N21 G0 Z100
N22 M30

```

Set the program datum point
 Load tool 1
 Start the spindle and move tool to starting point
 Feed tool to depth
 Move tool T0 the contour
 Set radius compensation RIGHT
 Activate the geometric calculations
 Move tool parallel to X-axis. Y-coordinate can be omitted.
 A linear movement under an angle. The starting point of this line is known, so programming the angle is sufficient
 The rounding between the intersecting lines of N9 and N11
 A linear movement under an angle to an end point.
 Axis parallel movements.

The linear movements under an angle including the rounding (N16) between these movements.

Cancel radius compensation
 Cancel geometric calculations
 Move the tool to a point free from the part
 Retract the tool in the tool axis
 End of the program

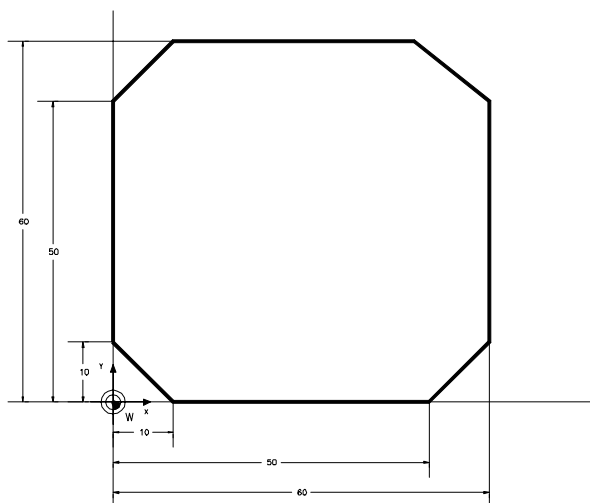
Notes: To get a graphical presentation of this program in one of the graphic modes of the control, you can add the following lines after N1:

```

N801 G98 X0 Y0 Z0 I100 J130 K-10
N802 G99 X10 Y10 Z0 I80 J105 K-10

```


Example 2 Insertion of a chamfer between linear movements



N64002

N1 G54

N2 S1000 T1 M6 (RADIUS 2 mm)

N3 G0 X-20 Y-20 Z10 M3

N4 G1 Z-10 F500

N5 G43 Y0

N6 G42

N7 G64

N8 X60

N9 I10

N10 Y60

N11 I10

N12 X0

N13 I10

N14 Y0

N15 I10

N16 X10

N17 G40

N18 G63

N19 Y-20

N20 G0 Z100

N21 M30

Refer to the corresponding lines in the first example.

Axis parallel movements

Chamfer between the linear movements of N8 and N10.

Axis parallel movements

Chamfer between the linear movements of N10 and N12.

Axis parallel movements

Chamfer between the linear movements of N2 and N14.

Axis parallel movements

Chamfer between the linear movements of N14 and N16.

Last movement to define the position of the chamfer

Cancel radius compensation

Cancel geometric calculations

Move the tool to a point free from the part

Retract the tool in the tool axis

End of the program

Notes: 1. To get a graphical presentation of this program in one of the graphic modes of the control, you can add the following lines after N1:

N801 G98 X-10 Y-10 Z0 I80 J80 K-10

N802 G99 X-5 Y-5 Z0 I70 J70 K-10

2. If a rounding should be programmed instead of the chamfer a few minor changes have to be made:

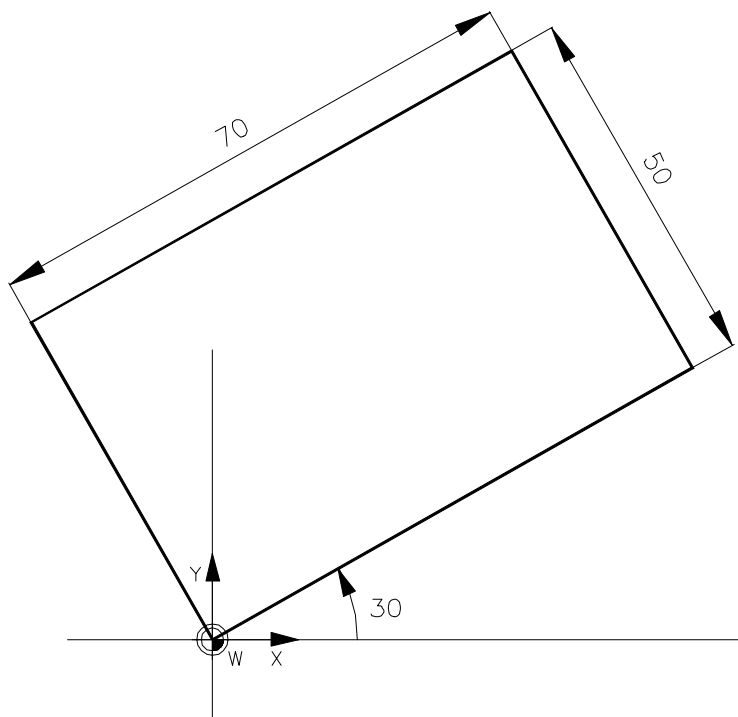
N9 G3 R10

N10 G1 Y60

N11 G3 R10

N12 G1 etc.

Example 3 Parallel lines



```

N64003
N1 G54
N2 S1000 T1 M6 (RADIUS 2 mm)
N3 G0 X-10 Y0 Z10 M3
N4 G1 Z-10 F500
N5 G43 X-4.33 Y-2.5
N6 G42
N7 G64
N8 X0 Y0 B1=30 I1=0

N9 X0 Y0 B1=120 I1=70

N10 X0 Y0 B1=-150 I1=50

N11 X0 Y0 B1=-60

N12 G40
N13 G63
N14 X-10 Y-10
N15 Z100
N16 M30
    
```

Refer to the corresponding lines in the first example.

Line defined by a support point (X0,Y0) and the angle with the X-axis

Line defined by a support point (X0, Y0), the angle with the X-axis and at a distance of 70 from the line through the origin.

Line defined by a support point (X0,Y0), the angle with the X-axis programmed in the direction of movement, and at a distance of 50 from the line through the origin.

Line defined by the endpoint (X0, Y0) and the angle with the X-axis programmed in the direction of movement.

Cancel radius compensation

Cancel geometric calculations

Move the tool to a point free from the part

Retract the tool in the tool axis

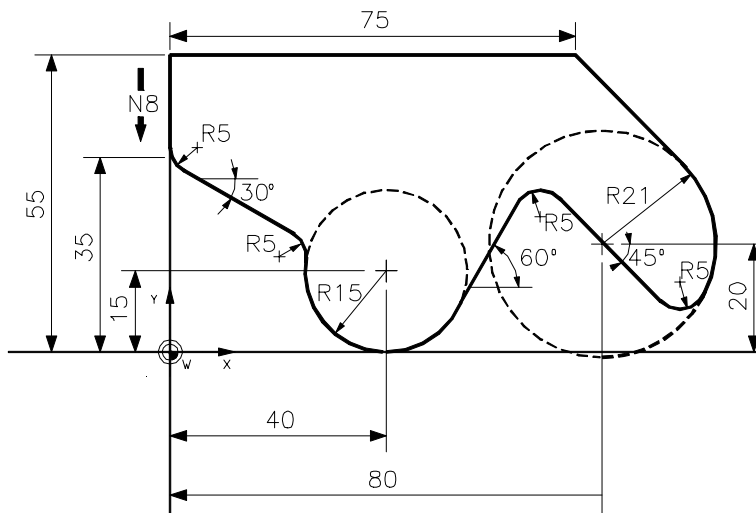
End of the program

Note: To get a graphical presentation of this program in one of the graphic modes of the control, you can add the following lines after N1:

```

N801 G98 X-35 Y-15 Z0 I110 J105 K-10
N802 G99 X-30 Y-10 Z0 I120 J95 K-10
    
```

Example 4 Line-to-circle and circle-to-line intersection



N64004
 N1 G54
 N2 S1000 T1 M6 (RADIUS 2 mm)
 N3 G0 X-5 Y60 Z10 M3
 N4 G1 Z-10 F500
 N5 G43 X0
 N6 G42
 N7 G64
 N8 B1=-90
 N9 G3 R5

 N10 G1 X0 Y35 I1=0 B1=-30 J1=2

 N11 G2 R5
 N12 G3 I40 J15 R15 R1=0
 N13 G1 B1=60
 N14 G2 R5
 N15 G1 X80 Y20 I1=0 B1=-45 J1=2

 N16 G3 R5
 N17 I80 J20 R21 R1=0
 N18 G1 X75 Y55
 N19 X-20

 N20 G40
 N21 G63
 N22 G0 Z100 M30

Refer to the corresponding lines in the first example.

Move tool downwards along the Y-axis

Make the rounding between the linear movements of N8 and N10

Move tool along the line. The starting point of this line is programmed as a support point, the angle is programmed in the direction of the movement and the right intersection point (J1=2) of the line and the circle of N12 should be used.

Make the rounding between the linear movement of N10 and the circular movement of N12

Follow the circle till the point of tangency between the circle and the linear movement of N13

A linear movement. The point of tangency is the known starting point.

A rounding between the linear movements of N13 and N15
 A linear movement through the centre of the circle of N17. The centre point is used as a support point of the line. The intersection point in the direction of movement should be used (J1=2)

Make the rounding between the linear movement of N15 and the circular movement of N17

Follow the circle till the point of tangency between the circle and the linear movement of N18

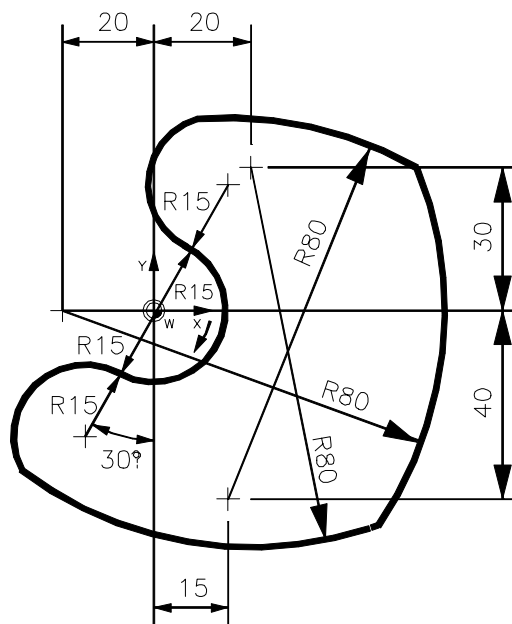
A linear movement to the programmed end point of the line
 Move tool parallel to X-axis, until the tool is free from the part

Cancel radius compensation

Cancel geometric calculations

Retract the tool in the tool axis and end of the program

Example 5 Circle-to-circle intersection



N64005

N1 G54 Refer to the corresponding lines in the first example.

N2 S1000 T1 M6 (RADIUS 2 mm)

N3 G0 X0 Y0 Z10 M3

N4 G1 Z-10 F500

N5 G43 X15

N6 G42

N7 G64

N8 G2 R15 R1=0

Move along the circle till the point of tangency with the next circle. The starting point of this circle is known from the previous blocks.

N9 G3 R15 B3=-120 L3=30 J1=1

Move along the circle defined by its polar centre point coordinates (B3=, L3=..) and the radius. The left intersection point (J1=1) with the circle in N10 should be used.

N10 I20 J30 R80 J1=1

Move along the circle defined by its Cartesian centre point coordinates and the radius. The left intersection point (J1=1) with the circle in N11 should be used.

N11 I-20 J0 R80 J1=1

The same type of movement as N10.

N12 I15 J-40 R80 J1=1

N13 R15 B3=60 L3=30 R1=0

Move along the circle till the point of tangency with the circle in N14. The circle is defined by its polar centre point coordinates (B3=, L3=..) and the radius.

N14 G2 X15 Y0 R15

Move along a circle programmed with end point and radius.

N15 G40

Cancel radius compensation

N16 G1 X0

Cancel geometric calculations

N17 G63

Move the tool to a point free from the part

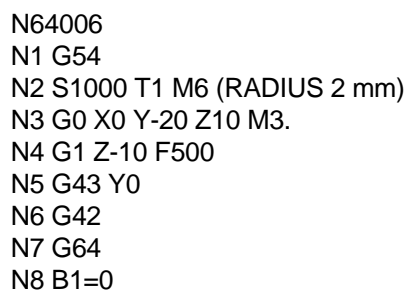
N18 G0 Z100 M30

Retract the tool in the tool axis. End of the program

Notes: To get a graphical presentation of this program in one of the graphic modes of the control, you can add the following lines after N1:

N801 G98 X-45 Y-60 Z0 I140 J140 K-10

N802 G99 X-40 Y-60 Z0 I125 J120 K-10



N9 G3 R40

A surrounding connecting circle between the line of N8 and the circle of N10. The line does not meet the circle
A circular movement till the point of tangency with the linear movement of N11. The circle is programmed with the Cartesian centre point coordinates and the radius.
A linear movement. The starting point of the line is the point of tangency with N10. The line does not meet the circle of N13.

A connecting circle between the line of N11 and the circle of N13.

A circular movement from the point of tangency with the circle of N12 till the point of tangency with the circle of N14. Similar to N12.

A linear movement to the point of tangency with the circle from N16.

A circular movement till the point of tangency with the surrounding connecting circle of N17

The surrounding connecting circle.

A linear movement along the X-axis to three programmed end point

Cancel radius compensation

Cancel geometric calculations

Retract the tool in the tool axis

End of the program

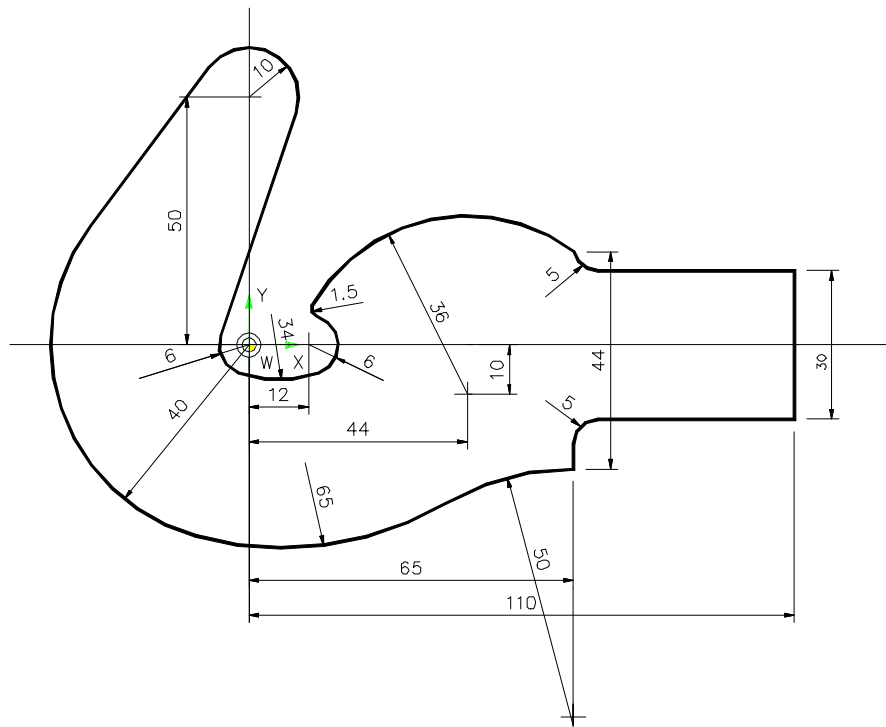
Note: To get a graphical presentation of this program in one of the graphic modes of the control, you can add the following lines after N1:

N801 G98 X-90 Y-15 Z0 I180 J90 K-10

N802 G99 X-70 Y-5 Z0 I140 J60 K-10

EXAMPLES

EXAMPLE 7 Connecting circle between circles



N1 G54

Refer to the corresponding lines in the first example. The radius compensation is set to be LEFT.

N3 G0 X120 Y-35 Z10 M3

N4 G1 Z-10 F500

N6 G41

N8 B1=180

N9 G3 R5
N10 G1 X65 Y-22 I1=0 B1=-90 J1=1

A rounding between the linear movements of N8 and N10
A downwards-linear movement parallel to the Y-axis till the

left intersection point of the circle from N11

A circular movement to the starting point of the connecting circle of N12

Connecting circle outside the circle of N11 and surrounding the circle of N13

Circular movement till the point of tangency with the line of the next block

Circular movement to the point of tangency with the line of the next block

Common tangent line between the circles of N15 and N17
Circular movement to the point of tangency with the

Connecting circle which surrounds the circles from N17 and

N19
Circular movement to the point of tangency with the

rounding of the next block. The circle intersects the circle of block N21

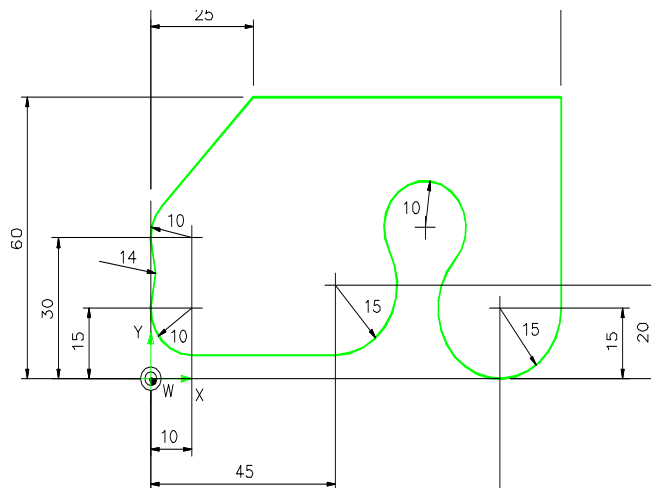
N20 G2 R1.5	The rounding between the two intersecting circles of N19 and N21
N21 G2 I44 J-10 R36 R1=0	Circular movement to the point of tangency with the line of the next block.
N22 G1 X65 Y22	Linear movement programmed with its end point
N23 B1=-90	A downwards linear movement parallel to the Y-axis
N24 G3 R5	A rounding between the linear movements of N23 and N25
N25 G1 X110 Y15 B1=0	Linear movement parallel to the X-axis programmed with end point and angle
N26 Y-40	Downward linear movement parallel to Y-axis
N27 G40	Cancel radius compensation
N28 G63	Cancel geometric calculations
N29 X120	Move the tool to a point free from the part
N30 G0 Z100 M30	Retract the tool in the tool axis and end of the program

Note: To get a graphical presentation of this program in one of the graphic modes of the control, you can add the following lines after N1:

N801 G98 X-50 Y-50 Z0 I170 J120 K-10
N802 G99 X-45 Y-45 Z0 I160 J110 K-10

EXAMPLES

Example 8 Connecting circle with greater are selection



N1 G54

Refer to the corresponding lines in the first example. The radius compensation is set to be LEFT.

N3 G0 X115 Y60 Z10 M3

N5 G43 X100

N7 G64

Notes

N13 G2 I10 J1

N14 G3 R14

N15 G2 I10 J

N16 G1 X25 Y60

N18 G40

N20 Z100

Note: To get a graphical presentation of this program in one of the graphic modes of the control, you can add the following lines after N1:

N802 G99 X-5 Y-5 Z0 I110 J70 K-10

Linear movement. The line is tangent to the circle of N9. The starting point of the line is known from the previous blocks.

Circular movement to the point of tangency with the connecting circle of the next block

Connecting circle between the circles of N9 and N11. The greater are is required. In this case the indicator K1=22 should be programmed. If this indicator is omitted the shorter are (K1 =12) is automatically chosen by the control.

Circular movement to the point of tangency with the common tangent line of N12.

Circular movement to the point of tangency with the

Connecting circle on the outside of the circles from N13 and

Circular movement to the point of tangency with the line of

Linear movement to the programmed end point.

Linear movement parallel to the X-axis

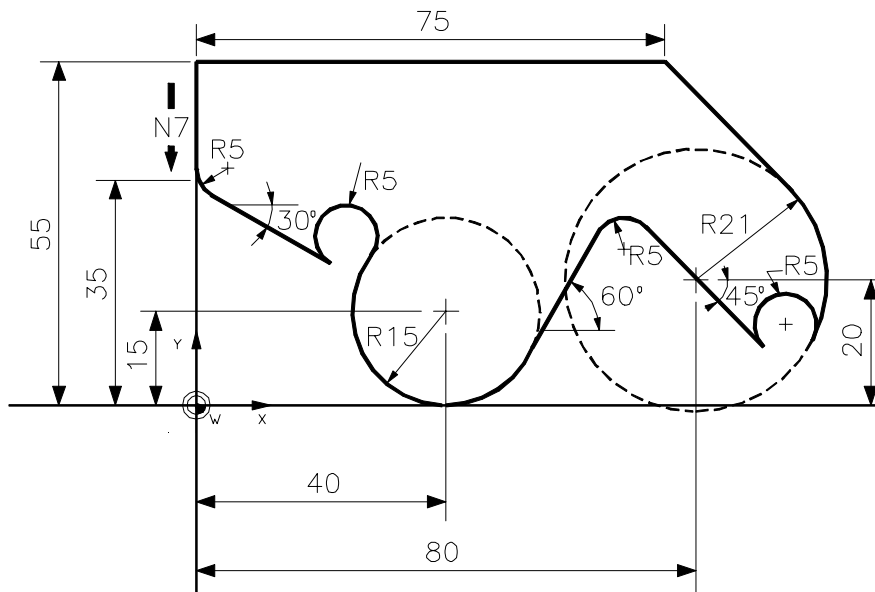
Cancel radius compensation

Cancel geometric calculation

Retract the tool in the tool axis

End of the program

Example 9 Non-continuous movement between line and circle



Compare this program with the one of example 4. The differences are:

N64009
 N1 G54
 N2 S1000 T1 M6 (RADIUS 2 mm)
 N3 G0 X-5 Y60 Z10 M3
 N4 G1 Z-10 F500
 N5 G43 X0
 N6 G42
 N7 G64
 N8 B1=-90
 N9 G3 R5
 N10 G1 X0 Y35 I1=0 B1=-30 J1=2
 N11 G2 R5 K1=4

The direction of rotation on the circle is changed and the indicator K1= 4 programmed to make the required circular movement.

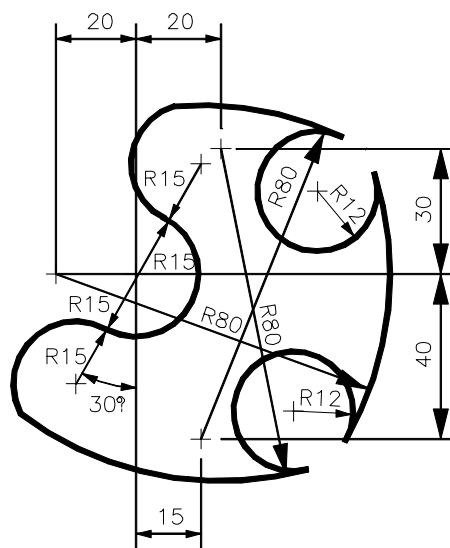
N12 G3 I40 J15 R15 R1=0
 N13 G1 B1=60
 N14 G2 R5
 N15 G1 X80 Y20 I1=0 B1=-45 J1=2
 N16 G2 R5 K1=4
 N17 G3 I80 J20 R21 R1=0
 N18 G1 X75 Y55
 N19 X-20
 N20 G40
 N21 G63
 N22 G0 Z100 M30

The same changes as in block N11.

Note: To get a graphical presentation of this program in one of the graphic modes of the control, you can add the following lines after N1:

N801 G98 X-10 Y-20 Z0 I140 J100 K-10
 N802 G99 X0 Y-10 Z0 I120 J80 K-10

Example 10 Non-continuous movement between two circles



Compare this program with the one of example 5. The differences are the roundings of the blocks N11 and N13. The indicator K1 =4 is programmed to make the required circular movement between the intersecting circles.

```

N64010
N1 G54
N2 S1000 T1 M6 (RADIUS 2 mm)
N3 G0 X0 Y0 Z10 M3
N4 G1 Z-10 F500
N5 G43 X15
N6 G42
N7 G64
N8 G2 R15 R1=0
N9 G3 R15 B3=-120 L3=30 J1=1
N10 I20 J30 R80 J1=1
N11 G2 R12 K1=4
N12 G3 I-20 J0 R80 J1=1
N13 G2 R12 K1=4
N14 G3 I15 J-40 R80 J1=1
N15 R15 B3=60 L3=30 R1=0
N16 G2 X15 Y0 R15
N17 G40
N18 G1 X0
N19 G63
N20 G0 Z100
N21 M30
    
```

Notes: To get a graphical presentation of this program in one of the graphic modes of the control, you can add the following lines after N1:

```

N801 G98 X-45 Y-60 Z0 I140 J140 K-10
N802 G99 X-40 Y-60 Z0 I125 J120 K-10
    
```

21. Appendix

21.1 Tilting of the operating plane

21.1.1 Introduction

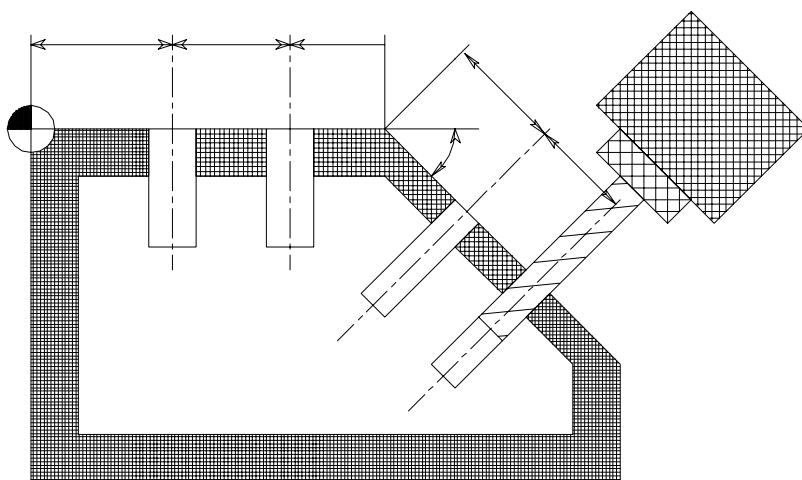
The control supports the tilting of operating planes on tool machines with tilting faces or tilting tables. Please consult your user handbook.

Typical applications, for example, are oblique drilling or contours, which lie obliquely in the operating area. In this way, the operating plane is always tilted about an active null point. Normally, the operation is programmed in a principal plane, e.g. X/Y plane; however, it is executed in the plane, which was tilted to the principal plane.

Consult the description of the G7 function for the programming of the freely programmable operating plane.

The tilting of the operating plane is defined and implemented using the G7 function. The G7 function is made up of two components:

- Definition of new operating planes, rotation of the coordinate system.
- In the event that it is programmed, tilt the tool vertically to the defined operating plane.



An operation on an oblique workpiece plane is programmed in local coordinates. In this way, the local X and Y coordinates lie in the oblique plane and the Z coordinate is fixed vertically in the plane. The machine recognizes the link between the local coordinates and the true machine axes and calculates this. The control calculates the tool correction factor.

Millplus **IT** distinguishes two machine types during tilting of the operating plane:

1) Machine with a tilting table

The location of the transformed machine axis is not changed with reference to the coordinates system fixed in the machine. If you rotate your table, and thus the workpiece, through 90°, for example, the coordinate system is not rotated through 90° with it. If, in the operating mode "Manual operation", you press the axis direction key Z+, the tool travels in the direction Z+.

2) Machine with a tilting face

The location of the tilted (transformed) machine axis is changed with reference to the coordinates system fixed in the machine:

If you rotate the tilting face of your machine and hence the tool, e.g. in the B axis about $+90^\circ$, the coordinates system is rotated with it. If, in the operating mode "Manual operation", you press the axis direction key Z+, the tool travels in the direction Z+ and X+ of the coordinates system fixed in the machine.

Using the G7 function you define the location of the operating plane by the input of tilt angles. The angles entered describe the angular components of a space vector.

If you program the angular components of the space vector, the control automatically calculates the angular position of the tilt axes. MillPlus **IT** calculates the location of the space vector and thus the location of the spindle axis, by means of rotation about the coordinates system fixed in the machine. The sequence of rotations for the calculation of the space vector is fixed: MillPlus **IT** turns the A axis first, next the B axis and finally the C axis.

The G7 function is effective from the start of its definition in the program.

MillPlus **IT** can only position-controlled axes automatically.

In the G7 definition, you can, in addition, input a safety distance to the tilt angles, with which the tilt axes are positioned.

Use only pre-set tools (full tool length in the tool table).

During the tilting process the position of the tooltip opposite the workpiece remains virtually unchanged (depending on the type of movement L1=).

MillPlus **IT** implements the tilting process using the power traverse.

21.1.2 Machine types

Milling machines with four or five axes can be used for the oblique machining of a workpiece.

Depending on the plane, which is tilted, other types of machine are needed for the working. At least two rotary axes and three linear axes are needed, in order to reach all sides and planes (except the under surface) without the need for remounting.

The possible types of machine are:

90° tilting face and turntable

The tilting face can be in two states. The upper and reverse sides can be worked by means of the tilting face. The four side surfaces can be worked using the turntable (C axis). The machine is only suited to all oblique operating planes if the tilting face can also be set in the oblique position manually.

Double turntable

The tables (A and C axes) are stacked. In this way, all sides and oblique operating planes can be worked.

Double turntable and 45° tilting face

The tables (A and C axes) are stacked. The A-axis is limited in its angular movement. In conjunction with the two tilting face states all sides and oblique operating planes can be worked.

45° double turntable

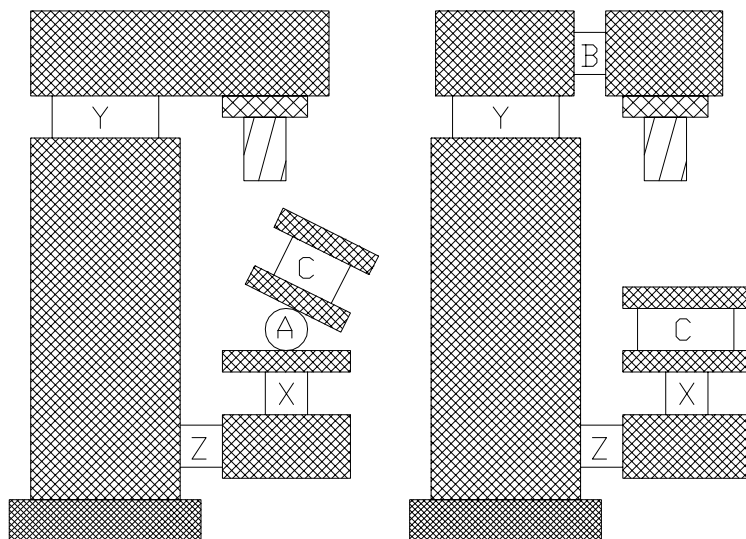
The tables (B and C axes) are stacked. The B axis is fixed in this way at an angle of 45° . All sides and oblique operating planes can be worked.

Turntable and rotating face

The face (B axis) can be freely positioned. In conjunction with the table (C axis) all sides and oblique operating planes can be worked.

Turntable and 45° rotating face

The face (B axis) is set at an angle of 45°. In conjunction with the table (C axis) all sides and oblique operating planes can be worked.

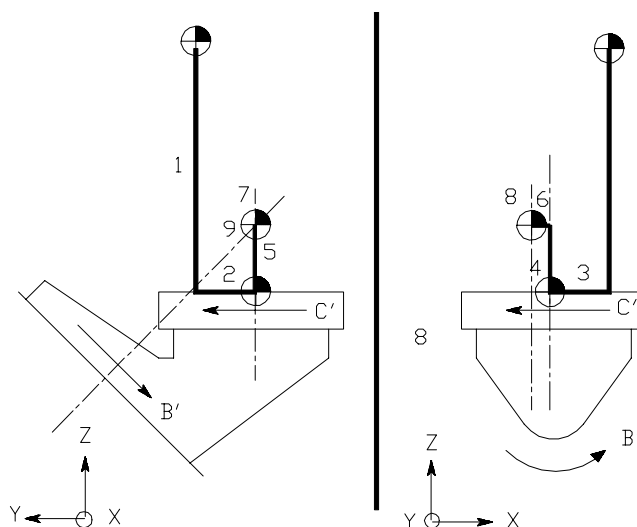


Outline of the most suitable machine types for use with oblique operating planes.

21.1.3 Kinematics model

The control needs a kinematics model of the machine in order to reset the programmed local coordinates in the oblique plane to the movements of the machine axes. A kinematics model describes the "construction" of the axes and the exact position of the different turning points on the rotary axes.

As an example, a kinematics model of the DMU 50 V machine is shown. The kinematics model is made up of a chain stretching from the workpiece to the machine frame. It is not necessary to describe the chain from the workpiece to the machine frame, because it includes no rotary axes.



Explanation of the drawing (for example kinematics model for the DMU 50 V

- 1,2,3 three elements in the X, Y, and Z directions in order to fix the (absolute) centre position of the workpiece table with reference to the marker positions.
- 4 element for definition of the C axis.
It is only necessary to describe the rotating axis of a rotary axis, not the centre point.
- 5,6 two elements in order to obtain the rotating axis of the second (incremental) rotary axis.
- 7 element for definition of the (incremental) direction of the second rotating axis. This direction is -45° in the A axis (all around the X axis).
- 8 element for definition of the B axis.
- 9 element in order to raise the -45° tilt (Element 7) again. In this way, the kinematics chain ends without rotation.

The kinematics model is entered by means of the machine settings MC600 to MC699.

A kinematics chain defines, by means of displacements and tilting, the way in which the rotary axes lie with respect to one another. Every displacement or tilting is determined as an element of the kinematics chain in three machine settings. Thus, a total of 25 elements of the kinematics chain can be determined. All rotary axes and positioning axes, which are present, should be described.

21.1.4 Operations

21.1.4.1 Manual operations

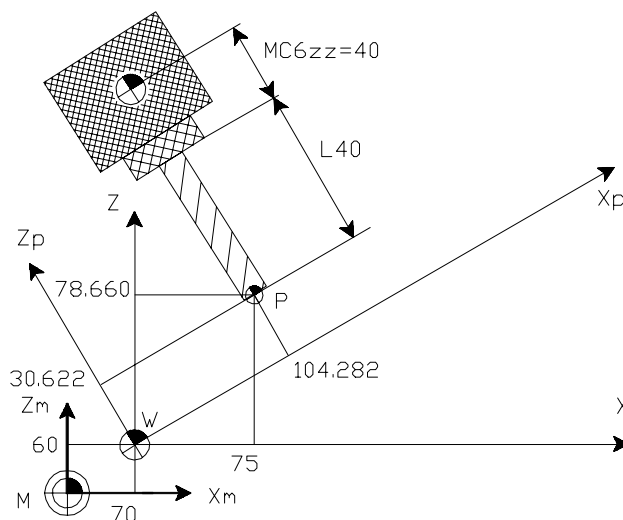
The axes are used along the local coordinates within the tilted G7 plane. E.G. Jogging skip of the Z-axis moves the tool vertically in the plane. All true linear machine axes can move in this way.

By means of a soft key, operation can be switched to the use of the true machine axes. The display then changes to show the true machine axes.

The user keys and the hand wheels for the linear axes can be assigned, according to choice, to the G7 plane or to the machine axes. The display is then implemented also in the G7 or machine axes plane. A new soft key in the soft key group <Step/continue> is used to implement the choice between G7 level or machine axes. For this purpose, this new soft key offers a "pause" option between the choices of jogging skip <advance> and <continue> soft key.

21.1.4.2 Display

If G7 is active, a yellow icon is displayed on the screen behind the tool number. By means of a small "p" on the right next to the "axis characters" an indication is given as to whether the display relates to the tilted operating planes or to the machine coordinates. The operating status has been enhanced with the current state of the programmed G7 space angle.



It is possible, using a new soft key in the "Jog operation type" soft key group, to change the display at the same time as the jog direction. If the position is displayed in machine coordinates, the position of the true tooltip is shown. See the next screen:

The position display on the screen can change between the position in the G7 plane (Xp, Zp) or in machine coordinates (X, Z).

Both are based on the active null point G52 + G54 + G92/G93.

21.1.4.3 Axis selection/position axis

An axis, which is not regulated, must be set to the correct position by hand. However, either before or after this, the oblique setting of the tool must also be entered by means of G7. Otherwise it will not be checked.

Comment: The expected position of the rotary axes is set parametrically in G7 using $n7=<\text{parameter number}>$. An axis selection or a positioning axis can be set manually using this information.

The axis selection or the positioning axis should also be followed in the kinematics model.

21.1.4.4 Reference point

If the reference point is approached during G7, the rotary axes remain in their reference position following the approach. The G7 plane is deactivated and the G17 plane is activated.

After running up the machine, but before approaching the reference point, the G7 plane is still active. After < reset CNC> the G7 plane is deactivated.

21.1.4.5 Intervention

If the G7 movement is interrupted, the exact position of the tooltip is displayed on the screen. Following interruption, the axes can be used in manual operation [mode].

Comment: Manual operation of the axis corresponds with the interrupted G7 plane (or, according to choice, the machine axes).

Following <Start> a move in position back to the interrupted point is affected. At the same time the axes run with positional logic corresponding with the G7 plane. Concurrently, the rotary axes rotate to the initial position.

21.1.5 Error messages

P306 Plane not clearly defined

The G7 plane is defined using a mix of absolute angles (A5=, B5=, C5=) and incremental angles (A6=, B6=, C6=).

Resolution: Use only absolute or incremental angles. Several G7 definitions with incremental angles can be defined, if necessary, one behind the other.

P307 Program plane not attainable

The G7 oblique setting defined cannot be attained, on account of the limited range of the rotary axes.

Resolution: Machines with a tilting face should tilt the face (by means of the M function) from the instantaneous position (horizontal or Vertical) following the other position.

O256 Machine type not recognized

The kinematics model in MC600 to MC699 is defining a type of machine, which is not supported for the oblique operating plane (G7). Only machine types with the following sequence of rotary axes, as viewed from the workpiece to the tool, are supported:

- A C
- C A
- C B
- C A fixed B -A fixed
(A fixed is a fixed tilt in the direction of the A axis, as, for example, the DMU50V has with -45°)
- C
- Axis change variants (C becomes B, and B becomes C) are also possible.

Resolution: The kinematics model should be entered in detail, with at least a description of those rotary axes present. The control must be run up once more.

21.1.6 Machine Constants

MC312 free operating plane (0=off, 1=on)

Activates the free operating plane. The G7 function can be programmed.

MC600 - MC699

There are 100 new machine settings (MC600 - MC699) for the description of the kinematics model. The model is described using a maximum of 25 elements, in which each element is described by means of four machine settings.

The following machine settings are used:

MC600	Kinematics chain (0=end, 1=tool, 2=workpiece)
MC601	Element (0,1=X, 2=Y, 3=Z, 4=A, 5=B, 6=C)
MC602	Element type (0=incremental, 1=absolute, 2=programmable)
MC603	Element insertion [°/mm degrees]
MC604, 608, 612, 616, 620, ..., 696	as MC600
MC605, 609, 613, 617, 621, ..., 697	as MC601
MC606, 610, 614, 618, 622, ..., 698	as MC602
MC607, 611, 615, 619, 623, ..., 699	as MC603

MC755 Free operating plane: rotation (0=coordinates cross, 1=axes)

If the desired rotation of the operating plane corresponds with the rotation of a rotary axis, the control has the choice between rotation using the relevant rotary axis or rotations using the coordinates cross. This choice is made with MC755.

E.G. on a machine with a (true) C axis the program instruction G7 C5=30 and MC755=0 produces a rotation of the coordinates cross over -30° and MC755=1 a rotation of the C axis over 30°

21.2 Look Ahead Feed (LAF) function

21.2.1 Introduction

The Look Ahead Feed function is used to carry out a precalculation on the programmed toolpath, while taking account of the dynamics of all axes involved. The toolpath speed is then adjusted to achieve the highest contour accuracy at the highest possible speed. However, the programmed feed is never exceeded.

Taking account of the programmed feed and the actual feed override setting; special high-performance algorithms ensure a homogenous feed for fast finishing times.

The execution speed of CAD-generated programs is substantially increased.

The user need not look at anything else when working with Look Ahead Feed.
This function cannot be influenced.

Only the G28 function was changed. The addresses for feed limitation were cancelled (see G27/628, starting from V320). Existing programs need not be adapted; they can be run as usual. These functions are ignored during machining operations. The machining operation may, however, continue.

During Look Ahead Feed the end point and centre point of a circle should match within 64 µm. In this case, the centre point is automatically corrected. As opposed to V310, there is no "compensation movement" at the end point. An error message is given if the end point and centre point do not match within 64 µm. This also applies to helix interpolation.

21.2.2 Detailed specification

1) Types of interpolation

The LAF function is active during:

- G0 Rapid traverse, including infeed movements
- G1 Feed movement
- G2, G3 Circle, including helix interpolation

The LAF function is inactive during:

- G6 Splines
- G74 Positioning movements
- G84 Threading
- G145, G45, G46 Measuring movement
- G182 Cylinder jacket interpolation with all permitted movements
- PLC-controlled axis movements (Home position)
- Auxiliary axis movements

2) Circular interpolation

- The circular accuracy achieved with LAF at higher speeds is higher than that with V3.10. This is true of circles made with G2/G3 and with cycles.

3) Previous contour accuracy functions (G28 function)

The following G28 programming functions are no longer active:

- I3=2 G1, G2, G3 with corner release distance (MC136)
- I3=3 G1 with programmable contour accuracy, (MC137) or I7=(0-10000)[μm]
- I4=2 G0 with corner release distance (MC136)
- I6=1 G2, G3 with feed limitation (MC135)

These functions are ignored during machining. The machining operation may, however, continue.

4) New error messages

P300 LAF: End point not on circle

Circle end point deviation exceeds 64μm.

The following applies additionally to a cylinder: 100 mm < R-cylinder < 10 m

Remedy: The end point should be defined more accurately.

P301 LAF: SW limit switch approached

The programmed path will go beyond the limit switches or outside + or - 100 m. In the case of straight lines this error is generated at the beginning of the wrongly programmed block. In the case of circles, it depends on the circular form and speed.

Remedy: The path should be defined within the possible range.

P302 No interpolation axis

The wrong axes have been defined for the selected type of interpolation:

No two main axes for circular interpolation

No rotary axis for cylinder jacket interpolation

Remedy: The missing axis should be defined.

INDEX

- ABSOLUTE PROGRAMMING297
 ACCELERATION REDUCTION..... 3, 86, 338
 APPLICATION TUNING CYCLE (G699) 3, 419, 420
 ARITHMETIC FUNCTION
 abs505, 506, 509, 510, 512
 arccos505, 509, 510
 arsin505, 509, 510
 artan247, 248, 505, 509, 510, 511
 ceil505, 507
 cosinus52, 505, 509, 510
 floor.....445, 448, 449, 455, 456, 457, 459, 460, 505, 507
 int (integer conversion)505, 506, 507, 510
 maximum505, 508
 minimum505, 508
 pi value505, 508
 Radians (rad)508, 509
 Reciprocals510
 sign505, 507
 sinus363, 505, 508, 509, 510
 sqrt505, 506, 510, 511, 512
 tangens505, 509, 510
 ARRAY
 add data to table359, 361, 362
 ARRnnnnn.CFG File365, 366
 Defining new table359, 360, 361, 362, 363, 364
 Deleting table359, 364
 Extract data from table359, 363, 365
 Filter an table359, 361, 362, 363
 Find value in table362
 Number of rows or column359, 361
 Sort a column in table359, 361, 364
 Tabel number365
 Test on existence of a table359, 361
 AXES CHARACTERS49, 229
 AXIS CHANGE632
 AXIS CONFIGURATION72, 74, 76, 265
 AXIS SELECTION631
 BASIC COORDINATE SYSTEM261, 275
 BASIC MOVEMENTS
 G0 Rapid traverse17
 G1 Linear interpolation20
 G2/G3 Circular interpolation27
 G78 Point definition178
 BOCK SEARCH226, 327, 334, 354, 356
 BTR6, 46, 326, 503
 CAD45, 82, 231, 232, 233, 237, 238, 240, 326, 497, 633
 CARTESIAN COORDINATES11
 CHIP BREAK421, 434, 435, 436, 438
 CLAMPING STATION9, 131, 134
 CLIMB MILLING196, 200, 204
 CONFIGURATION FILE352, 357, 365
 CONSTANT CUTTING FEED (F1=) ... 100, 101, 337, 475, 476
 CONTOUR ACCURACY86
 CONTROL PANEL5, 6, 131, 133, 134
 CONVENTIONAL MILLING317
 COOLANT225, 227, 228, 369, 371, 372
 COORDINATE SYSTEM7, 9, 21, 101, 104, 131, 134, 253, 255, 262, 263, 266, 267, 268, 269, 282, 627
 COORDINATES CROSS50, 632
 CORNER ACCURACY46
 CUTTING FORCE MONITOR498
 CYCLE
 G77 Bolt hole circle174
 G79 Activate cycle180
 G81 Drilling cycle183
 G83 Deep hole drilling cycle185
 G84 Tapping cycle188
 G85 Reaming cycle191
 G86 Boring cycle193
 G88 Groove milling cycle199
 G89 Circular pocket milling cycle203
 CYCLE DESIGN 3, 4, 465, 473
 DATA COMMUNICATIONS PROGRAM474
 DEFINING COORDINATES11
 DRILLING CYCLES
 G781 Drilling434
 G782 Deep-hole drilling435
 G783 Deep-hole drilling with additional chip break438
 G784 Tapping440
 G785 Reaming442
 G786 Boring443
 G790 Back-boring451
 EDIT MODE291
 EDITING7, 290
 EMERGENCY STOP226
 END POINT WINDOW292
 ETHERNET LINK474
 FEEDRATE FUNCTION
 G25/G26 Enable/Disable feed- and/or speed-override... 83
 G27/G28 Positioning functions84
 G4 Dwell time40
 G94/G95 Select feedrate unit.. 15, 145, 215, 267, 519, 559
 F-FUNCTION13
 FMS TOOL MEMORY252, 255, 499
 FORK HEAD G316228
 G106 KINEMATIC SETTLEMENT OFF221
 G108 KINEMATIC SETTLEMENT ON222
 G11
 One point geometry63, 66, 67
 Two line geometry63, 66, 69
 Two point-geometry63, 66, 68
 G125 LIFTING TOOL ON INTERVENTION
 OFF3, 224, 225, 226, 340
 G126 LIFTING TOOL ON INTERVENTION
 ON3, 224, 225, 226, 340
 G136 SECOND AXES CONFIGURATION FOR FORK HEAD
 ON3, 227, 228, 229, 230
 G137 SECOND AXES CONFIGURATION FOR FORK HEAD
 OFF3, 227, 228, 229, 230
 G141 3D-TOOL CORRECTION231
 G141 3D-TOOL CORRECTION
 Dynamic TCPM233
 Nominal tool231
 Surface normal vector232, 233, 235, 237
 TCPM231, 232, 233, 235, 236, 237, 238, 239
 Tool Centre Point Management232
 Tool vector233, 236, 237, 238, 239
 G153 CANCEL C154 ZERO POINT SHIFT .3, 256, 257, 258, 340, 341
 G154 ACTIVATE ZERO POINT DISPLACEMENT 3, 256, 257, 258, 340, 341
 G174 TOOL WITHDRAWAL MOVEMENT259
 G180 BASIC COORDINATE SYSTEM261
 G182 CYLINDRICAL COORDINATE SYSTEM263
 G2/G3 CIRCULAR INTERPOLATION
 Angle of circular arc29
 CCW14, 27, 175, 176, 332, 432, 482
 Circular arc32, 137, 163, 168, 174, 181
 Circular interpolation14, 163, 633
 CW14, 27, 175, 176, 332, 432, 482, 486, 516
 G217 DEACTIVATE ANGULAR HEAD3, 321, 323
 G218 ACTIVATE ANGULAR HEAD3, 321, 322, 323, 341
 G300 PROGRAMMING ERROR MESSAGES330
 G301 ERROR IN A PROGRAM THAT JUST HAS READ IN331
 G303 M19 WITH PROGRAMMABLE DIRECTION332
 G310 STORE TABLE ON DISK 3, 329, 333, 334, 335
 G311 LOAD TABLE FROM DISK 3, 329, 333, 334, 335
 G318 READ PALLET OR JOB TABLE DATA 3, 329, 336
 G319 QUERY ACTUAL TECHNOLOGY DATA336

INDEX

G320 QUERY ACTUAL G DATA	337
G321 QUERY TOOL DATA	342
G322 QUERY MACHINE CONSTANT MEMORY	343
G324 QUERY G-GROUP	344
G325 QUERY M-GROUP	345
G326 QUERY ACTUAL POSITION	346
G327 QUERY OPERATION MODE	347
G331 WRITE TOOL DATA	348
G341 CALCULATION OF THE G7-SPACE ANGLE	329, 350, 351
G350 DISPLAY WINDOW	354
G351 WRITE TO FILE	356
G39 ACTIVATE/DEACTIVATE TOOL OFFSET	90
G6 SPLINE-INTERPOLATION	41
G61 TANGENTIAL APPROACH	136
G62 TANGENTIAL EXIT	140
G66/G67 DETECT TOOL DIRECTION	16, 18, 77, 162, 181, 223, 389
G699 APPLICATION TUNING CYCLE	3, 419, 420
G74 ABSOLUTE POSITION	171
G8 TILTING TOOL ORIENTATION	54
Compensation movement	56, 57, 236, 237, 633
GENERAL PROGRAMMING INFORMATION	5
GEOMETRIC CALCULATIONS	14, 161, 561, 562, 602
Circle and line	562, 598, 611
Concentric circles	562, 601
Connecting circle indicator	562, 602
Intersection point	98, 147, 149, 159, 562, 563, 565, 566, 584, 586
Intersection point indicator	149, 159
Line and circle	562, 596, 610
Parallel line	158, 618
Tangency indicator	151
Two circles	562, 599
Two lines	157, 417, 511, 562, 563, 566, 608
GEOMETRY FUNCTIONS	
G63/G64 Geometric calculations	15, 144, 308, 309, 311, 316, 559
G72/G73 Scaling or mirror imaging	15, 166, 211
G9 Define pole position	58
GRAPHIC	
G195 Graphic window definition	270
G196 End contour description	272
G197/G198 Begin inner/outer contour description	273
G199 Begin contour description	279
G98 Graphic window definition	218
G99 Definition of workpiece blank as a box	220
HEAD POSITION CONTROL G642	416
HELIX	22, 27, 33, 101, 137, 141, 145, 302, 305
HELIX INTERPOLATION	33, 101, 145, 302, 305
HIGH-PERFORMANCE ALGORITHMS	633
IMPROVEMENTS	3
INCH	5, 14, 164, 353, 468, 559
INTRODUCTION	1, 13, 285, 352, 359, 389, 422, 465, 513, 627, 633
JOG OPERATION	49, 631
KINEMATIC MODEL	21, 57, 216, 222, 223, 227, 228, 235, 478
LASER	
G642 Temperature compensation	3, 367, 389, 416, 417
LASER MEASURING	
G600 Calibration	367, 369, 370, 377, 416, 489
G601 Measure tool length	367, 371
G602 Measure length and radius	367, 372
G603 Check of individual edge	367, 374
G604 Tool breakage control	367, 375
G615 Turning tool measurement	367, 388, 528, 559
G951 Calibration	4, 367, 463
G953 Measure tool length	4, 367, 463
G954 Measure length, radius	4, 367, 463
G955 Cutter control shank	4, 367, 463
G956 Tool breakage control	4, 367, 463
G957 Cutter control shape	4, 367, 463
G958 Tool setting length, radius, corner radius	4, 367, 463
LOOK AHEAD FEED (LAF)	84, 85, 633, 634

MACHINE BUILDER HANDBOOK	345, 502
MACHINE CONSTANT . 14, 21, 113, 116, 223, 226, 227, 265, 292, 334, 377, 378, 380, 381, 409, 411, 468, 491, 514	
MACHINE REFERENCE POINT	8
MAGNETIC DIGITAL CASSETTE	5
MAIN AXIS COORDINATES	166
MAIN PLANES	
G17 Main plane XY	72
G18 Main plane XZ	74
G19 Main plane YZ	76
G7 Tilting working plane	47
MANUALS	1, 3, 15, 81, 223, 226, 258, 327, 330, 331, 333, 334, 335, 354, 367, 463, 495, 627, 628, 630, 631
MEASURING	
Air blow	108
Basic measurement movement	117, 121
Calibration mandrel	369
Calibration ring	115, 116, 488
Calibration tool	369, 370, 378, 416
Infrared probe	489
Measuring distance	53, 107, 108, 113, 114, 116, 391, 393, 394, 395, 397, 400, 402, 404, 406, 407, 409, 411, 412
Square-head probe	110, 111
MEASURING CYCLES	
G620 Angle measuring	389, 392, 393
G621 Position measuring	389, 394
G622 Corner outside measuring	389, 395, 396
G623 Corner inside measuring	389, 397, 398
G626 Datum outside rectangle	389, 399, 400
G627 Datum inside rectangle	389, 401, 402
G628 Circle measurement outside	389, 403, 404
G629 Circle measurement outside	389, 405, 406
G631 Plane measurement	389, 390, 407, 408
G633 Measuring zero point of 2 holes	3, 389, 409, 410
G634 Measuring zero point of 4 holes	3, 389, 411, 412
G640 Rotary table center offset	3, 389, 413, 415, 416
MEASURING FUNCTIONS	
G145 Linear measuring movement	241
G148 Read probe status	250
G149 Read tool data or zero offset	251
G150 Change tool data or zero offset	254
G45 Axis parallel measuring movement or tool dimensions	106
G46 Measuring a full circle or probe calibration	112
G49 Checking on tolerances	117
G50 Processing measuring results	121
M-FUNCTION . 14, 53, 108, 145, 189, 227, 243, 246, 345, 491	
MILLING CYCLES	
G700 Face turning	346, 421, 424, 425
G730 Multipass milling	426
G787 Pocket milling	445
G788 Key-way milling	447
G789 Circular pocket milling	449
G797 Pocket finishing	455
G798 Key-way finishing	457
G799 Circular pocket finishing	459
MIRRORING	168, 169, 170, 212, 302, 305, 339
NESTING	289, 291, 292
NESTING LEVEL	289, 291
OFFSET	
Tool length	90, 338
Tool radius	90, 338
OFFSET PROGRAMMING	90
OPERATING MANUAL	8, 111
OPERATION MODES	
G70/G71 Inch/Metric programming	14, 164, 559
G90/G91 Absolute/incremental programming	14, 44, 58, 206, 208, 234, 267, 308, 309, 311, 559
OPERATOR MACHINE CONSTANTS	367
OVERRIDE	
Feed	15, 83, 233, 440, 453, 517
Speed	83, 440, 453
PALLET	129, 336
PITCH OF HELIX	33, 34
PLANE FEED	476

- PLANE TILT G748
 POLAR COORDINATES 11, 18, 20, 32, 71, 207, 208, 273, 275, 280
 POSITION AXIS631
 POSITIONING LOGIC 18, 85, 86, 180, 338, 422
 PROBE CALIBRATION 16, 112, 115
 PROGRAM BLOCK 6, 71
 PROGRAM CONTROL
 G14 Repeat function70
 G22 Macro-call78
 G23 Main program call81
 PROGRAM WORD5, 13
 PROGRAMMING ACCURACY4
 RADIUS COMPENSATION
 G240 Contour check OFF4, 325, 326, 328
 G241 Contour check ON3, 4, 99, 325, 326, 327, 328
 G40 Cancel tool radius compensation92
 G41/G42 Tool radius compensation95
 G43/G44 Tool radius compensation to/past endpoint ... 103
 REFERENCE POINT631
 SCALING46, 131, 134, 146, 167, 169, 339, 389
 SCALING FACTOR168
 S-FUNCTION14, 480
 SOFTWARE LIMIT SWITCH8, 259, 260, 270
 SPACE VECTOR628
 SPARE TOOL252, 255, 495, 496, 497, 500, 502
 SPEED SUPERIMPOSE SWITCH373, 374
 SPINDLE SPEED RANGE480, 491
 SPLINE
 Bezier-Splines41, 42, 44
 Cubic splines41
 Radius of curvature46
 Spline16, 41, 42, 46
 STANDARD CONFIGURATION265
 S-WORD500
 SYNCHRON GRAPHICS45, 218, 270, 271
 TABLE POSITION CONTROL G640413
 TAPPING15, 188, 190, 421, 440, 453
 Floating tap holder188
 Retap thread189
 T-FUNCTION14, 495
 THREAD MILL CUTTER39
 TOOL CHANGE52, 228, 302, 305, 484, 495, 496, 497
 TOOL CORRECTION56, 242, 521, 525
 TOOL DIMENSIONS73, 75, 77, 122, 168, 500
 TT130
 G606 Calibration378
 G607 Measuring tool length379, 386
 G608 Measuring tool radius381
 G609 Measuring tool length and radius ..367, 377, 383, 489
 G610 Tool breakage control385
 G611 Turning tool measurement387, 526, 559
 Stylus115, 369, 377, 386, 526
 TURNING
 G227 unbalance monitor OFF324, 531, 559
 G228 unbalance monitor ON324, 531, 559
 G302 Overwriting radius compensation parameters332
 G33 Treadcutting in turning47, 89, 517, 518, 559
 G36/G37 Activate/deactivate turning mode16, 89, 513, 515, 559
 G96/G97 Constant cutting speed217, 520, 559
 Tool orientation233, 342, 349, 499, 502, 515, 521, 524, 527, 528, 535, 536, 537, 538, 539, 540, 541, 542, 543, 544, 545, 546, 547, 548, 549, 550, 551
 TURNING CYCLE
 G691 Unbalance measurement419, 514, 530, 532, 534, 559
 G692 Unbalance check419, 514, 530, 534, 559
 G822 Clearance axial461, 535, 536, 538, 557, 559
 G823 Clearance radial461, 535, 537, 539, 558, 559
 G826 Clearance axial finishing461, 535, 538, 559
 G827 Clearance radial finishing461, 535, 536, 537, 539, 558, 559
 G832 Roughing axial461, 535, 540, 542, 559
 G833 Roughing radial461, 535, 541, 543, 559
 G836 Roughing axial finishing461, 535, 542, 559
 G837 Roughing radial finishing461, 535, 540, 541, 543, 559
 G842 Grooving axial461, 535, 544, 548, 559
 G843 Grooving radial461, 535, 545, 549, 559
 G844 Universal grooving axial roughing3, 461, 535, 546, 550
 G845 Universal Grooving radial roughing3, 461, 535, 547, 551
 G846 Grooving axial finishing461, 535, 544, 548, 559
 G847 Grooving radial finishing461, 535, 545, 549, 559
 G848 Universal Grooving axial roughing3, 462, 535, 546, 547, 550
 G849 Universal Grooving radial roughing. 3, 462, 535, 551
 G850 Undercut (DIN 76)3, 462, 535, 552
 G851 Undercut (DIN 509 E)4, 462, 535, 553
 G852 Undercut (DIN 509 F)4, 462, 535, 554
 G861 Treadcutting cylinder4, 462, 535, 555, 556
 G862 Treadcutting taper4, 462, 535, 556
 TURNING MODE. 89, 217, 278, 323, 324, 332, 387, 388, 514, 524, 559
 UNBALANCE
 Concentric accuracy530
 Unbalance324, 362, 363, 364, 419, 514, 530, 531, 532, 534, 559
 UNIVERSAL POCKET
 Finishing macro290, 292
 G201 Start contour pocket cycle296
 G202 End contour pocket cycle305
 G203 Start pocket contour description307
 G204 End pocket contour description309
 G205 Start island contour description310
 G206 End pocket contour description313
 G207 Call island contour macro315
 G208 Quadrangle contour description317
 Islands287, 306, 311, 312, 313, 314, 316, 320
 Pseudo-Island286
 VERSION V410138, 141
 VERSION V420234, 469, 507
 VERSION V5103, 325
 VERSION V5113
 VERSION V5203, 325
 WINDOW SIZE172, 173
 WIREPLOT GRAPHIC274, 281
 WORDWISE206, 208
 WORKPIECE ZERO POINT3, 58, 61, 130, 133, 256, 257, 316, 513
 ZERO POINT SHIFT
 Extended MC84>0121, 130, 133, 251, 254
 G51/G52 Cancel/Activate pallet zero point shift15, 129, 130, 211
 G53/G54 Cancel/Activate zero point shift15, 130, 559
 G92/G93 Incremental/absolute zero point shift9, 15, 31, 32, 45, 131, 134, 167, 169, 209, 210, 229, 236, 253, 261, 265, 266, 302, 305, 339, 390, 559, 631
 Standard MC84=0121, 130, 251, 254, 331